eta/VPG

VIRTUAL PROVING GROUND

CAE TOOLS FOR VEHICLE SYSTEM APPLICATIONS

eta/VPG™
User's Manual

A Mechanical System Simulation Software

eta/VPG™ Version: 3.4

Manual Release Date: Apr 08, 2009

FOREWORD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Engineering Technology Associates, Inc.
1133 East Maple Road, Suite 200
Troy, MI 48083

Phone:(248) 729-3010
Support:(800) ETA-3362
Fax:(248) 729-3020
Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/VPG are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of their respective owners.

Copyright 2009 Engineering Technology Associates, Inc. All rights reserved

TABLE OF CONTENTS

CHAPTER 1: INTRODUCTION

Documentation	Section 1.1
eta/VPG History	Section 1.2
eta/VPG Methodology	Section 1.3
eta/VPG Software Package	Section 1.4
eta/VPG Special Features	Section 1.5

CHAPTER 2: FEATURES AND FUNCTIONS OVERVIEW

General	Section 2.1
Pre Processor	Section 2.2
Post Processor	Section 2.3
Menu System	Section 2.4
Function Keys	Section 2.5
Display Window	Section 2.6
Icon Bar	Section 2.7
Mouse Functions	Section 2.8
Keyboard Entry	Section 2.9
Specifications	Section 2.10
Line Data	Section 2.11
Conventions	Section 2.12
VPG File Menu	Section 2.13
Recommended Naming Convention	Section 2.14
LS-DYNA Cards	Section 2.15
Local Coordinate System	Section 2.16
Entity Selection	Section 2.17

CHAPTER 3: GETTING STARTED

Opening/Creating an eta/VPG Database File	Section 3.1
Setting Up a VPG Model	Section 3.2

CHAPTER 4: MAIN MENU

File	Section 4.1
Parts	Section 4.2
Pre	Section 4.3
Road	Section 4.4
Suspension	Section 4.5
Tire	Section 4.6
Safety	Section 4.7
ALE-FSI	Section 4.8
DTM	Section 4.9
Heeds	Section 4.10
Analysis	Section 4.11
Fatigue	Section 4.12
Post	Section 4.13
Utilities	Section 4.14
View	Section 4.15
Macro	Section 4.16
Help	Section 4.17

CHAPTER 5: FILE MANAGER

New	Section 5.1
Open	Section 5.2
Restart	Section 5.3
Save	Section 5.4
Save As	Section 5.5
Merge Database	Section 5.6
Import	Section 5.7
Export	Section 5.8
Print	Section 5.9
Print Setup	Section 5.10
Exit	Section 5.11

CHAPTER 6: PRE-PROCESSOR

Line	Section 6.1
Surface Options	Section 6.2
Element Options	Section 6.3
Node Options	Section 6.4
Model Checker	Section 6.5
Set Menu (LS-DYNA)	Section 6.6
Set Menu (NASTRAN)	Section 6.7
Boundary Conditions (LS-DYNA)	Section 6.8
Boundary Conditions (NASTRAN)	Section 6.9
Material Property (LS-DYNA)	Section 6.10
Material Property (NASTRAN)	Section 6.11
Element Property	Section 6.12
Contact Interface	Section 6.13
Dyna Miscellaneous	Section 6.14
Super Element (NASTRAN)	Section 6.15
Mass Menu (LS-DYNA)	Section 6.16
Assembly (LS-DYNA)	Section 6.17
Mass Menu (NASTRAN)	Section 6.18
Assembly (NASTRAN)	Section 6.19

CHAPTER 7: VPG MODULES

Road Menu	Section 7.1
Suspension Menu	Section 7.2
Tire Model	Section 7.3
Safety Module	Section 7.4
Occupant Safety	Section 7.5
Dummy Positioning	Section 7.6
Delete Model	Section 7.7
ALE-FSI Module	Section 7.8
DTM Module	Section 7.9
Heeds	Section 7.10

CHAPTER 8: ANALYSIS

Case Control	Section 8.1
Analysis Submission	Section 8.2

CHAPTER 9: POST PROCESSING

Introduction Section 9.1
General Overview Section 9.2
File Manager Section 9.3
Icon Bar Section 9.4
File Format Section 9.5
Configuration File Section 9.6
File Menu Section 9.7
Edit Menu Section 9.8
Tool Menu Section 9.9
Option Menu Section 9.10
Post Process Section 9.11
Graph Section 9.12

CHAPTER 10: UTILITY

Viewing Options Section 10.1
Icon Bar Section 10.2
Part Control Section 10.3
Utility Section 10.4
Display Options Section 10.5
Macro Section 10.6

APPENDICES

APPENDIX A: VPG Hardware and Software Requirements
APPENDIX B: Supported IGES Entity Types
APPENDIX C: RADIOSS / LS-DYNA / NASTRAN Conversion

Chapter 1

Chapter 1: Introduction

1.1 Documentation

The eta/VPG Documentation consists of Manuals and Tutorials. This manual serves as the primary description of all eta/VPG functions and provides the user with a description of each of these functions and the necessary mechanics of how to use them. The Tutorials demonstrate how to implement the various functions in eta/VPG for use in specific finite element modeling applications.

This documentation was created to complete the following objectives:

1. Provide a description of general-purpose pre- and post-processor functions for constructing FEA models and reviewing analysis results.
2. Provide descriptions of the usage of utility modules for meeting VPG-specific application requirements. Examples of this are tire model generation, road surface library usage, suspension template usage, fatigue life prediction calculation, and signal processing.
3. Describe methods to interface with external data; reading and writing files to various MCAE solvers and accepting data from commercial CAD software.

Accompanying this manual is a set of Tutorials specifically designed to introduce modeling and application topics to the user. The Tutorials provide examples to demonstrate the different techniques used in creating a VPG simulation and how to use various modeling functions. The Tutorials also serve as a training guide for new users to gain experience with VPG techniques, methodology, and software.

In addition to the eta/VPG Manuals and Tutorials, the solver-related features pertaining to eta/VPG's dynamic simulations can be found in the LS-DYNA User's Manual. The LS-DYNA User's Manual is the documentation for the LS-DYNA software, developed by Livermore Software Technology Corporation (www.lstc.com).

1.2 The History of eta/VPG

The eta/VPG software has its origins in the automotive CAE community, where the need for efficient finite element modeling tools emerged in the mid to late 1980's. These tools were needed to address the drive for shortened vehicle development schedules and the desire to implement CAE simulations in the design process.

ETA had been an early innovator in the area of finite element pre & post processing,

developing and commercializing the eta/FEMB software in the mid-80's. This software met the needs of the automotive CAE engineers and was quickly adopted to help the transformation of the automotive CAE community.

As the need for more complex simulations arose, there was a simultaneous increase in low-cost, high-speed computing capacity. This provided the opportunity to create system-level simulations of events, mimicking the test process more accurately. Prior to this time, conventional CAE techniques consisted of smaller, specific analyses tailored to a specific task (such as NVH), separate from the global system-level analysis of the automobile.

The eta/VPG developers were faced with the need for an integrated analysis that included both component-level and full vehicle applications with the ability to perform real-time simulations. The analysis would be dynamic and nonlinear -- unlike the static, linear analysis techniques utilized by the then-current CAE practices.

ETA first tested this concept on simple mechanisms, simulating mechanical systems such as engine/connecting rod/piston/crankshaft systems and suspension mechanisms. The mechanisms were simulated as a dynamic, nonlinear system in real-time events. The results demonstrated that the motion and forces derived from the VPG simulation were the same as the rigid body linkage motion simulation results produced from conventional multibody dynamics software. However, when implementing flexible bodies (FE), the benefit of obtaining realistic stress and strain results of flexible components from the VPG simulation in an event-based fashion produced valuable analysis data that could not be generated in traditional FEA approaches.

ETA engineers moved another step forward by simulating a rotating tire impact in a pothole event. A tire model was developed to include a control volume technique to simulate the tire air pressure behavior under an impact condition. The simulation of the frictional contact of the tire model with the rigid road surface was extremely realistic and correlated with existing tire data.

To complete the system-level simulation, the need for dynamic test-based boundary conditions was necessary. ETA therefore formed an agreement with MGA Research Corporation to model the company's vehicle proving ground facility located in Burlington, Wisconsin. ETA created road surface profiles that could be digitized from the proving ground construction drawings in a CAD surface format. FEA mesh could be generated from the CAD surface data and produced a selection of specific road profiles representing the characteristics of the proving ground testing conditions.

After years of development and evaluation, it has been confirmed that VPG technology produces repeatable, reliable, and correlated analysis results. The level of confidence from users and correlation results has lead to aggressive simulation use and a broad application scope.

Dr. John Hallquist, at Livermore Software Technology Corporation (LSTC) has been instrumental in the development of the VPG application technology by implementing various features and functions of LS-DYNA to meet VPG requirements. This partnership with LSTC has allowed the VPG methodology to grow in scope of application and reliability.

The release of VPG 1.0 in 1998 was an effort to address the specific needs of system modeling by combining the tools created for tire modeling and suspension modeling with the finite element meshing and associated tools found within the eta/FEMB product.

As usage progressed and the needs of users expanded, fatigue analysis software was embedded into VPG, as were the signal processing tools used in NVH post processing.

The dual nature of many of the models created in VPG and the desire to construct one model for use in several types of simulations drove the development of the VPG/Safety module. This module was incorporated into eta/VPG version 2.0, which was released in 2002.

1.3 VPG Methodology for Event Simulations

Virtual Proving Ground (VPG) is a general term used to reference a simulation methodology using the tire and proving ground road surface approach.

The VPG method is a set of techniques used with an explicit, nonlinear, dynamic analysis program, which allows for the complete analysis of a mechanical system, including all joints, bushings, materials, and geometric non-linearities using an event-based analysis. The class of problems targeted is those in which a mechanical system is to be analyzed in a dynamic sense. In other words, when a mechanical system is in use, the displacements, forces, accelerations, and stresses occur in real-time. The VPG method allows for the calculation of all of these quantities simultaneously, using a single analysis run.

While VPG methodology was initially developed for full vehicle simulations based on a proving ground durability cycle, the concept can also be applied to other dynamic mechanical systems.

1.4 eta/VPG Software Package

eta/Virtual Proving Ground (eta/VPG) is a fully integrated, dynamic, nonlinear, finite element software package used to create, analyze, edit, and visualize dynamic, nonlinear engineering problems. The software includes an integrated preprocessor, post processor, and solver. VPG is a complete CAE software toolset for applying theory and engineering principles common in areas of mechanical and structural engineering. eta/VPG provides a single package for use in analysis of multi-body dynamics problems, linear static, nonlinear static, and dynamic nonlinear finite element analysis.

eta/VPG's strength lies in its ability to integrate problems that are treated as multidisciplinary by other software packages. eta/VPG allows the user to combine multi-body dynamics problems with structural finite element analysis problems, providing real-time kinematics or dynamics as well as the stress or strain response of the structure in real time.

eta/VPG consists of three primary modules: the VPG/PrePost module, the VPG/Structure Module and the VPG/Safety Module. Each module may operate independently, and usage is controlled via licensing options.

VPG/PrePost is a general-purpose, full-featured modeling software for the construction and results display of finite element models. This module allows users to import CAD data, construct CAD data, import existing FE models, and construct the geometry and mesh required for finite element analysis. All boundary conditions and material properties may also be created in this module. Output of the model in various finite element solver formats is available.

VPG/Structure contains many special features designed specifically for system level or full vehicle analysis. These features allow for easy modeling of joints and bushings, finite element meshes, boundary conditions, materials, properties, suspension system components, tire models, and road surface models.

VPG/Safety was introduced in release 2.0 of eta/VPG. This module is targeted at LS-DYNA users who wish to construct vehicle impact and occupant safety simulations. This module is

based on two integral sub-components: a crash tool library and the test procedures associated with a specific governmental regulation. The Crash Tool Library contains the dummy or barrier needed to perform a specific analysis. For instance, a European Side Impact test requires an EEC Side Impact Barrier and a EUROSID dummy model. These are combined with the testing procedure (vehicle velocity, etc.) and measurement criteria to establish a process. VPG/Safety follows that process to set up the simulation model. A total of 21 different governmental or insurance institute test procedures are contained within VPG/Safety.

1.5 eta/VPG Special Features

eta/VPG was initially designed with unique features specifically for virtual proving ground simulations. These features are not found in competing general purpose pre- and post-processors. The development of these features and functions was necessary to satisfy the following VPG requirements:

1. Generation of complicated full vehicle/full system models and retention of complex non-linear material properties, contact definitions, etc.
2. Generation and retention of user-defined libraries and modules such as a tire library, proving ground road surface libraries, etc.
3. Post-processing of potentially large amounts of analysis data (displacement, forces, acceleration, stress, and strain) derived from time domain solution of multiple events and the conversion from time domain results into frequency domain results (frequency, mode shape and PSD, etc.).
4. Post-processing of stress and strain results of multiple events and conducting the fatigue life prediction.
5. Modeling and set-up of vehicle impact analyses using FMVSS, ECE, and Insurance Institute standard testing procedures.

The key features are briefly described below. A more detailed explanation of these features is found in Chapter 7.

LS-DYNA 971 Interface Module

eta/VPG incorporates a complete, direct LS-DYNA version 971 interface. eta/VPG reads and writes all LS-DYNA 971 cards, eliminating the need for text editing of the input deck. It creates and retains all material nonlinear properties, contact definitions, and loading conditions.

Tire Model Generator

eta/VPG's tire model generator allows for easy construction of tire models. The tire model generation tool employs specific tire geometry and inflation pressure to automatically construct a three-dimensional finite element model. Tire models are used for both vehicle durability and NVH applications.

Specialized tire models are available for users requiring detailed models of tires. These models are suitable for use in detailed, tire-focused analysis and require detailed material information.

Suspension Model Generator

eta/VPG's sophisticated suspension model generator automates the FEA modeling of the most popular suspension types. Suspensions may be modeled using flexible, finite element representations or rigid members. Material properties of the suspension component can be

specified using the nonlinear stress-strain characteristics of the material. Included in eta/VPG's suspension libraries are these suspension types:

- *MCPHERSON H-ARM*
- *TRAILING ARM*
- *MCPHERSON A-ARM*
- *HOTCHKISS*
- *SOLIDAXLE*
- *TWIST BEAM*
- *FIVE LINK*
- *HONDA 5-LINK*
- *QUADRA LINK*
- *SHORT LONG ARM*

Road Surface Library

eta/VPG'S road surfaces form a full vehicle durability evaluation platform ready for kinematics and stress analyses of component, subsystem, and vehicular models. Road surfaces are generated using any 3D data or selected from the VPG library. The Library contains digitized models of the following MGA Proving Ground road surfaces:

POTHOLE TRACKS
ALTERNATE ROLL SURFACE
COBBLESTONE TRACKS
BODY TWIST LANE
RIPPLE TRACKS
WASHBOARDS
CHATTERSTRIP
PAVE SURFACE

Fatigue Analysis Program Module

eta/VPG's built in fatigue post processor automatically performs a fatigue life analysis prediction to identify the key damage events and stress amplitudes. This data is then used to calculate the percentage of fatigue life remaining at the completion of the durability cycle. eta/VPG displays these results in easily read, combined fatigue life contour plots.

Signal Processing Module

eta/VPG performs full vehicle NVH studies on simulated proving ground surfaces. eta/VPG automatically converts time-domain analysis results into frequency-domain via FFT (Fast Fourier Transform). eta/VPG determines both low and mid range frequencies up to 250Hz for operating mode shapes, frequencies, structural and airborne noise, and frequency responses such as idle shake, rough road, power train, and wheel imbalance.

Chapter

2

Chapter 2: An Overview of eta/VPG Features

This chapter provides an overview of the main features of eta/VPG. Detailed descriptions of the use of these features are provided in subsequent chapters.

2.1 GENERAL

The eta/VPG Processor has a complete graphical user interface that is operated on Windows (Windows NT and above) and UNIX/Linux-based workstations including IBM, HP, SUN, and SGI operating systems. The model generation, input file preparation, and results processing activities are all performed in an identical environment. The solution can be executed on both local and/or remote server systems.

2.2 PRE PROCESSOR

eta/VPG's extensive Preprocessing capabilities contain all of the functions necessary for expedient, high quality, finite element modeling. Users can read in data with VPG's CAD interface, build their model from scratch, read in an existing model, or a combination of the approaches.

CAD INTERFACE

eta/VPG enables users to read in CAD geometry data from the following CAD systems:

- Catia Version 4
- Catia Version 5
- Unigraphics version 18
- Unigraphics NX to NX5
- Proe
- Parasolid
- Solidworks
- Sat
- STEP
- IGES
- Inventor

The IGES interface reads files generated from any CAD system. Supported IGES entity types include:

Name	Type
Null Entity	0

Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	149
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

For the German automotive industry, VDA file format is supported. Supported VDA entity types include:

POINT
CIRCLE
CURVE
SURF
CONS
FACE

If CAD data is not available, eta/VPG has a complete geometry and surface building capability, which acts as an integrated CAD data generator. This is detailed in Chapter 6.

EXTENSIVE MODEL BUILDING FUNCTIONS

eta/VPG includes a comprehensive selection of functions for creating and modifying line data and CAD surfaces. In addition to automeshing functions, element generation through 2-line, 4-line, 6-line and 8-line mesh creates beam, shell, and solid elements.

COMPREHENSIVE MODEL MODIFICATION FUNCTIONS

All aspects of the model may be modified using the pre-processing menus. All defined material, elements, nodes, contacts and analysis parameters may be modified, deleted, copied, etc., as required by the user.

FULL SYSTEM ASSEMBLY

The ASSEMBLY menu allows users to create assemblies which may be modified as a group. This includes part replacement and connectivity attributes of the model.

AUTOMESHING

eta/VPG's automeshing function easily eliminates 90% of the time required to mesh trimmed and standard surfaces. The automeshing function creates quadrilateral elements with a minimum of triangular elements.

Various automeshing algorithms are available, including Triangular Meshing, Paving Mesh and Topology Automeshing. A tetra element automesher is available, generating both 4-noded and 10-noded solid elements.

MATERIAL/ELEMENT PROPERTIES

eta/VPG supports all LS-DYNA and NASTRAN material and element property cards. eta/VPG's ability to create and assign material and element properties directly on a displayed model greatly reduces the amount of editing required. Definition cards appear at specific junctions in the session prompting the user for material/element properties.

A Material Library is available in the LS-DYNA environment, allowing users to select common ferrous and aluminum materials from a pre-populated set of material definitions.

Users also have the option to specify the analysis software (NASTRAN, LS-DYNA, RADIOSS, PAMCRASH) prior to assigning properties.

CONTACT INTERFACE

eta/VPG seamlessly interfaces with LS-DYNA, allowing the user to create and assign impact, sliding, or automatic interfaces for VPG applications. eta/VPG displays contact properties in easily read and modified CONTACT CARDS with a high degree of speed and flexibility.

BOUNDARY CONDITIONS

eta/VPG's BOUNDARY CONDITIONS menu allows the user to quickly create and verify constraints and loads on VPG models.

MODEL INTEGRITY CHECKING FUNCTIONS

The functions in eta/VPG's MODEL CHECKER menu quickly validate models for element orientation, size, skew, connectivity, and interior angles. Model validation default values are easily adjusted to suit the user's needs.

CONSTRAINTS

eta/VPG supports all LS-DYNA Constraint Cards for quick, easy definition and manipulation of joints, welds, rivets, etc.

2.3 POST-PROCESSOR

eta/VPG integrates a general-purpose, complete post-processor for data post processing of analysis result files. Analysis results are seamlessly imported from VPG's LS-DYNA and NASTRAN solvers to the post processor for quick, easy interpretation of analysis results. Once entered, VPG's full complement of post processing functions allow the user to graphically display

and manipulate simulation result files with contour and deformation animation, contour plots and fills, and geometry deformation.

VPG's post-processing functions provide the user with numerous ways of animating and viewing the analysis results. The user has the options of animating select frames, viewing single frames, altering the time step, rotating the model, or viewing only select parts of the model.

The post processing function automatically prompts the user for the result file when activated. VPG post processes the following types of result data:

D3PLOT (d3plot01, etc.)
DYNA DEFORMED GEOMETRY (.defgeo)
HISTORY (.his)
NASTRAN PUNCH (.pch)
NASTRAN OUTPUT2 (.op2)
NASTRAN PACKED PUNCH (.pac)

VPG provides an option to create a more compact post-processing file. This binary file, called a “pp” file due to its extension “ *.pp ”, is created after reading the result file into eta/VPG. Once the .pp file is created, the user may read this file to save both disk storage space and time when reloading the analysis results. The .pp file requires less space than the analysis result file used to create it and can be used for all subsequent post processing functions. It also requires significantly less time to load the .pp file.

FATIGUE

VPG allows the user to generate and post process fatigue result files from within the VPG Interface.

ANIMATE CONTOUR

This function is used to map the stresses, strains, and strain energy of the model across time. Animations can be edited to even, odd, or specified frames.

ANIMATE DEFORMATION

This command allows the user to animate displacements within the model in real time. Animations can be edited to even, odd, or specified frames.

CONTOUR FILL

This function maps stresses, strains, and strain energy in the model for a single step. It allows the user to view the contour values by superimposing a fill-color contour image onto the model. The values are displayed in a color legend in the upper right hand corner of the screen.

CONTOUR LINES

This function, similar to the CONTOUR FILL function, allows the user to check the model's contour values for a single step. A color, wire frame, contour-line plot of the component result is superimposed onto the current model.

DEFORMED SHAPE

This command displays the displacement results of the model for a single step. The undeformed model shape is displayed in white. The model can be animated to show the transition between the undeformed and deformed model.

ELEMENT STRESS

This function displays the stress results of each individual element as opposed to the contour plot functions, which display the stress results in terms of the average stress at each node. A color legend for the corresponding color values is displayed in the upper right hand corner of the screen.

TIME HISTORY PLOT

eta/VPG's TIME HISTORY PLOT functions enable the user to visualize the results of an analysis with XY plots. VPG offers a wide range of tools to manipulate the information on the display screen with labels, colors, multiple graphs, and a host of advanced filtering techniques e.g., FIR, Butterworth, SAE, scaling, smoothing, and averaging.

FFT (FAST FOURIER TRANSFORM)

The TIME <--> FREQUENCY function allows the user to convert time domain to frequency domain response for signal processing analysis. The FFT equation is used to convert time domain to frequency domain and frequency domain to time domain for any graph plot.

2.4 MENU SYSTEM

The program starts in the MAIN menu (see figure 2.4.1) and branches out into submenus. The user selects a sub-menu by mouse pick or keyboard entry. Descriptions for these menu options are located in their respective sections.

Figure 2.4.1 Main Menu

FILE	Imports and exports data to and from VPG.
PART	Creation, display control and manipulation of part entities.
PRE	Contains a menu of preprocessing functions.
LINE	Creates and modifies line/surface data.
SURFACE OPTIONS	Creates and modifies surfaces in VPG.
ELEMENT OPTIONS	Creates and modifies elements.
NODE OPTIONS	Creates, copies, transforms, and manipulates nodes.
MODEL CHECKER	Checks element criteria (warpage, boundary, aspect ratio, etc.).
SET MENU	Creates node and element sets for Super element files substructure files, etc.
BOUNDARY CONDITIONS	Creates and verifies constraints and loads on a finite element model.
MATERIAL	Creates and assigns material properties.
ELEMENT PROPERTY	Creates and assigns element properties.
CONTACT INTERFACE	Creates and modifies sliding/rigid wall interfaces for LS-DYNA.
DYNA MISCELLANEOUS	Handles distinct LS-DYNA miscellaneous data.
NASTRAN MISCELLANEOUS	Handles distinct NASTRAN miscellaneous data.
SUPER ELEMENT	Assigns and modifies SUPER ELEMENT ID numbers (only for NASTRAN application).
MASS MENU	Creates and modifies MASS.
ASSEMBLY	Arrange and assemble parts' or part sets' connections.
CONSTRAINT	Creates and Modifies constraints
LOAD CARDS	Creates, defines and modifies LOAD CARDS (LS-DYNA)
INITIAL CONDITION	Creates, defines and modifies INITIAL CONDITION CARDS (LS-DYNA)
ROAD MENU	Defines road surfaces.
SUSPENSION MENU	Defines and auto-generates front and rear automotive suspensions.
TIRE MODEL	Defines and auto-generates tire models.
SAFETY	Analyzes vehicle impact and occupants' safety.
ALE-FSI	Sets up ale models especially for explosion analysis
DTM	Sets up a series of drop tests for a specimen
HEEDS	Define Heeds parameters for optimization
ANALYSIS	Analyzes proving ground events.

FATIGUE	Analyzes element fatigue.
POST	Provides options for viewing the results of an analysis. Plots dynamic characteristics of the structure vs. time, velocity, etc. in X-Y Curve format
UTILITIES	Provides a series of VPG "tool kit" functions.
VIEWING OPTIONS	Manipulates the display, position, and perspective of the model.
MACRO	Records commands executed by VPG and replays commands in automated fashion

2.5 FUNCTION KEYS

Function keys 1 through 8 act as shortcuts the most frequently used menus. The F1 (Function key 1) is reserved for the Main Menu.

F1	Main Menu	**F5**	Model Checker
F2	Element Options	**F6**	Node Options
F3	File Open	**F7**	Surface Options
F4	Line	**F8**	Pre-Processor

2.6 DISPLAY WINDOW

VPG breaks the screen into six distinct regions. The regions are used to receive input or display messages for the user. The six regions are illustrated on the following page. See Figure 2.6.1.

DRAWING WINDOW -- Model and definition cards are displayed in this area.
MENU BAR – The groups of menus containing all eta/VPG modules.
ICON BAR-- This group of commands dynamically manipulates the display, position, and perspective of a model.
MENU -- Commands and the Command Options are displayed in this area. They can be accessed via the keyboard mouse.
DIALOGUE WINDOW -- VPG displays comments and messages to the user and accepts keyboard entry commands in the dialogue window.
DISPLAY PARAMETER OPTIONS WINDOW -- These commands set the plot options for the current model.

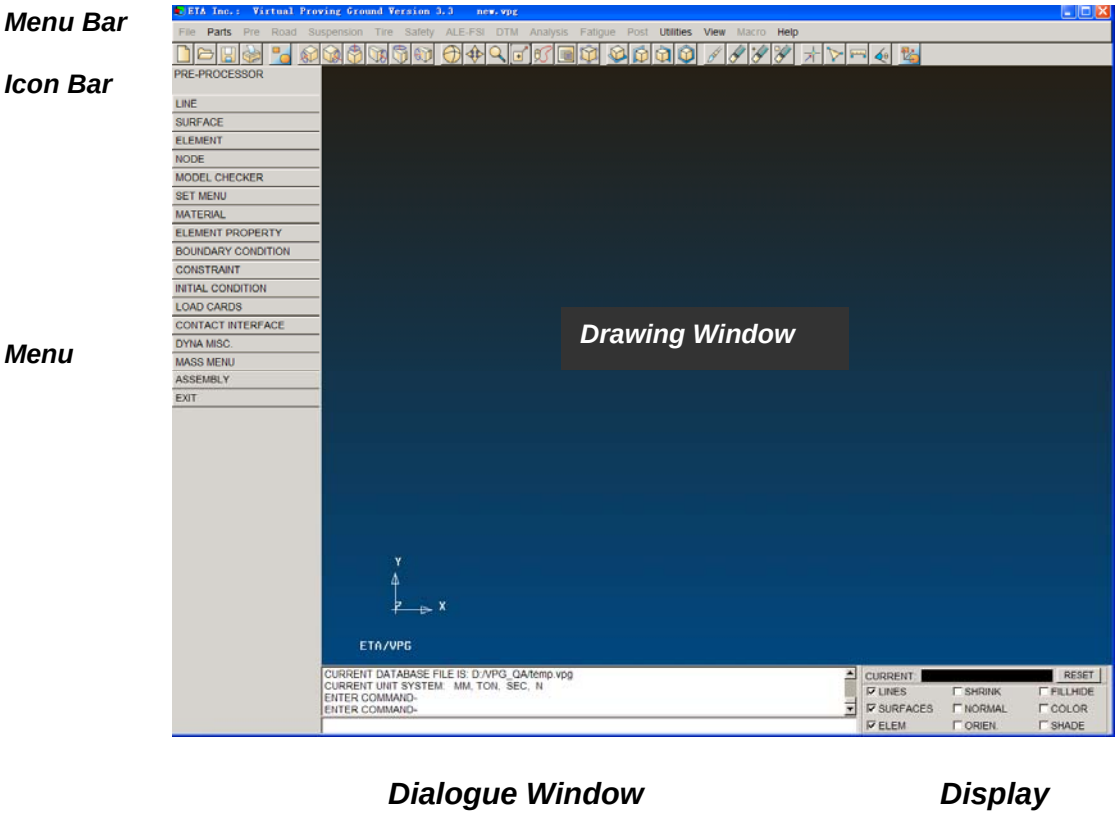


Figure 2.6.1 VPG Window Layout

2.7 ICON BAR

The icon bar is designed to give the user easy access to the most commonly used functions in eta/VPG. Some of the functions represented in the icon bar are also located in the different menus. The user may simply click these icons to activate the functions instead of clicking through the menus.

NEW
Creates a new database file.

OPEN
Opens a database.

PRINT
Creates a postscript file of the display area and sends the file to the printer (default) or to a file. Prior to printing, the postscript driver must be initialized.

PART ON/OFF
Turns the selected parts on or off. The **PART TURN ON/OFF** dialog window is displayed once selected.

VIRTUAL X ROTATION

The displayed model will dynamically rotate about the global X-axis when the cursor is moved up or down.

VIRTUAL Y ROTATION

The displayed model will dynamically rotate about the global Y-axis when the cursor is moved up or down.

VIRTUAL Z ROTATION

The displayed model will dynamically rotate about the global Z-axis when the cursor is moved up or down.

SCREEN X ROTATION

The displayed model will dynamically rotate about the screen X-axis when the cursor is moved up or down.

SCREEN Y ROTATION

The displayed model will dynamically rotate about the screen Y-axis when the cursor is moved up or down.

SCREEN Z ROTATION

The displayed model will dynamically rotate about the screen Z-axis when the cursor is moved up or down.

Note: Screen axis rotation uses the display screen as a plane of rotation. The screen X-axis is horizontal. The screen Y-axis vertical and screen Z-axis is perpendicular to screen X and Y.

FREE ROTATION

This function is a combination of SX and SY. Moving the mouse up/down manipulates SX. Moving the mouse left/right manipulates SY. Moving the mouse diagonally combines the movements of both commands. Clicking the left mouse button stops the rotation. This function can also be activated by pressing Control and Left mouse button, and exits by releasing left mouse button.

PAN

This command enables the user to translate the model by following the movement of the cursor. If the cursor is moved off the screen, the cursor reappears at the center of the screen. Clicking the left mouse button stops the pan. This function can also be activated by pressing Control and Middle mouse button, and exits by releasing Middle mouse button.

CURSOR ZOOM

The user picks a point about which to zoom. The model is centered about this point and the user may move the cursor up or down to zoom in or out. This function can also be activated by pressing Control and Right mouse button, and exits by releasing Right mouse button.

Note: If the cursor is moved off the screen in functions ROTATE, PAN or CURSOR ROOM, the cursor re-appears the opposite of the screen automatically.

WINDOW ZOOM

The user defines the corners of the zoom window by positioning the cursor on the display screen. The user presses the left mouse button and drags the cursor diagonally down until the desired window size is reached. After releasing the left button, the section included in the window is displayed in full screen.

FREE HAND ZOOM

The user defines a free region by pressing the left mouse button and dragging the cursor on the display screen. Release the left button, and the section included the region is displayed in full screen.

ACTIVE WINDOW

This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. The user defines the region by dragging a window over the desired area. eta/VPG displays the elements, lines, and surfaces within the window (volume in space) as active. Other objects on the screen remain masked and inactive.

FILL

Rescales the model to include all entities belonging to parts that are currently turned on. FILL automatically zooms in or out until the model fits the viewing area of the screen.

TOP VIEW

Automatically displays the model from the TOP or in the XY-plane.

SIDE VIEW

Automatically displays the model from the SIDE or in the XZ-plane.

REAR VIEW

Automatically displays the model from the REAR or in the YZ-plane.

ISOMETRIC VIEW

Automatically displays the model from the isometric plane (60-degree isometric).

CLEAR

Removes the highlighted entities from the screen.

REDRAW MODEL

eta/VPG updates the screen after each command. Occasionally, the use of specific commands requires updating of the image with an additional step (e.g., when operating the dynamic zoom at the same time that the element normal are displayed, it is necessary to activate the REDRAW command to resize the arrows that represent the element normal).

CLEAR NODE
Clear the display of connecting nodes

CLEAR ELEMENT
Clear the display of connecting elements

IDENTIFY NODE
This function enables the user to identify any node.

IDENDFIY ELEMENT
This function enables the user to identify any element.

DISTANCE BETWEEN TWO NODES
This function enables the user to calculate the distance between two nodes.

ANGLE BETWEEN THREE NODES
This function enables the user to measure the angle between two vectors formed by three nodes.

COORDINATES ON/OFF
This function enables the user to turn on/off coordinate systems.

2.8 MOUSE FUNCTIONS

All VPG model rotation/pan/zoom functions are accessible via selection using the left mouse button. To access a function, the user selects the desired button using the mouse pointer and presses the left mouse button. This button is also used for selecting definition cards, locating the cursor in definition cards, creating drag windows, locating points, nodes, elements, etc. The right mouse button activates a floating, pull-down menu with commonly used model manipulation functions (See Figure 2.8.1). All functions on the menu will be described in following chapters.

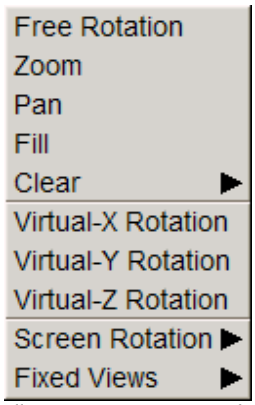


Figure 2.8.1 Floating Pull Down Menu accessed via Right Mouse Button

2.9 KEYBOARD ENTRY

To increase speed and efficiency, all VPG functions can also be accessed by keyboard entry. Entering a one or two-letter combination followed by the return key activates each command of the menu that the user presently has on the screen. For main menus, the letter combination is the first two letters of a one-word command or the first letter of each of the first two words of a two or more word command. As the user types the keys, the matching command will be highlighted. For example, the keystroke entry for the command ELEMENT OPTIONS/ SURFACE MESH in the above menu would be "s", "m" followed by the RETURN key. For the ELEMENT OPTIONS/ MODIFY command, the keystroke entry would be "m", "o" followed by the RETURN key. For control keys, the user need only type the first letter to access the function.

2.10 SPECIFICATIONS

The standard version of VPG has the following specifications for model databases:

Entity	Maximum Count
LINES	150,000
POINTS	1,500,000
SURFACES	98,000 6,500,000 Edge Points 400,000 Control Points
GRIDS	5,000,000
ELEMENTS	5,000,000
PROPERTIES	1,000
PIDS	5,000

2.11 LINE DATA

VPG's built-in translator converts and filters line data from the following programs into a neutral line format:

IGES (lines and surfaces)
DXF file formats
VDA file

There are additional external translators to support CAD files generated from the following programs:

CATIA4
CATIA5
Proe
Solidworks
STEP
Unigraphics versions 18 and NX to NX5

2.12 CONVENTIONS

This manual is designed to reduce the amount of reading material on the page and maintain text clarity. Several fonts and symbols are implemented throughout the manual. An example is given at the bottom of the page.

FIXED FONT **-**This font indicates text found within VPG e.g., menu
(ALL CAPS, BOLD) **-** Names, subsections, commands, and options within commands, etc.
Proportional Font - This font indicates explanatory text e.g., command descriptions, notes, and section titles.

ALL CAPITALS - This font indicates a function, menu name, card, command, etc. found in explanatory text.
> - The greater than symbol directs the user to read the text displayed in VPG's DIALOGUE WINDOW.
? - The question mark directs the user to select an option that is listed in VPG's MENU WINDOW.
• - The bullet signals a description of the previous command or situation.

-CREATE.
4 - PLATE ELEMENT

This section covers the options for the PLATE ELEMENT subsection of CREATE ELEMENTS.

1. VPG prompts:
> **PICK NODES/POINTS FOR ELEMENT**
? **NODE**
POINT
• To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
• An element will be created after three or four nodes/points are selected.
• EXIT or ABORT will exit this function.

2.13 VPG OPEN FILE WINDOW

The VPG OPEN FILE WINDOW allows users to access files and directories. The DIRECTORIES and FILES windows are accompanied. VPG OPEN FILE WINDOW is a convenient tool for viewing of directories and files.

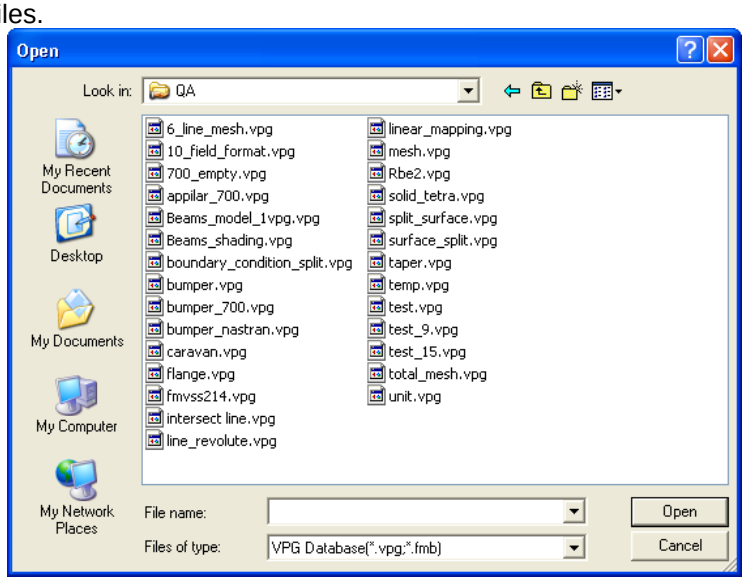
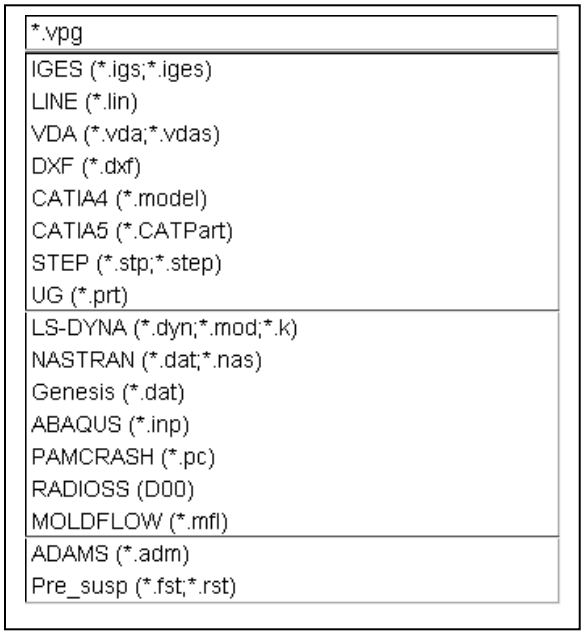


Figure 2.13.1 Open File Window

All files listed in the OPEN FILE window are displayed from the current directory based on a filter (filename extension such as **.vpg**). The current directory is listed at the top of the LOOK IN drop-down menu. Changing the filter parameter in the File Type drop-down menu sets the file type filter. The FILE NAME window allows the user to enter the file name and directory manually.

2.14 RECOMMENDED NAMING CONVENTION (.his, .lin, .bin, etc.)

The protocol for naming files during a VPG session includes attaching suffixes to the file names that specify the file types. Examples of suffixes include:



Example: When reading in a line data file, VPG prompts for a line data file name (all file Names in that directory with the suffix **.lin** are listed in the options area). The user then selects the appropriate file name. This practice makes the file name selection convenient and organizes the user's work directory.

2.15 LS-DYNA CARDS

eta/VPG supports all LS-DYNA input data from versions 971 and earlier. As a result, models generated using eta/VPG require no external editing to complete the file for execution. The input "cards" are arranged as they are in the LS-DYNA manual. eta/VPG displays the necessary input parameters in windows as shown in Figure 2.15.1 and as described below.

CARD SELECT BUTTON

The CARD SELECT BUTTON displays the number of definition cards and allows the user to jump conveniently to the desired card. The card button is not an option when only one card is needed to define the selected property or the number of cards is dependent on user-defined values.

CARD DESCRIPTION FIELD

The CARD DESCRIPTION FIELD allows the user to enter a name for the defined properties.

EDITING FIELDS

VPG's smart editing field only allows values within the specified range to be entered. If the value entered is beyond the LS-DYNA defined range VPG prompts the user when the user tries to move to another field or card.

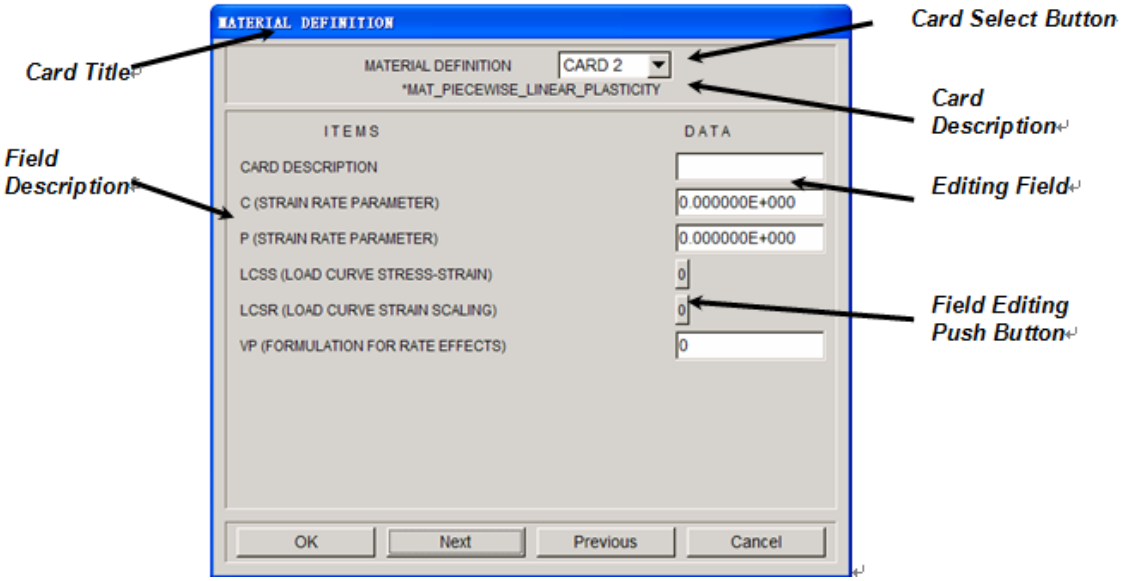


Figure 2.15.1 Card Definition Window

FIELD DESCRIPTION

The field description indicates the value to be added to the editing field. If the field description text is gray, it indicates that the value entered in that field is dependent on another field that must be edited first.

EDITING PUSH BUTTON

An editing button forwards the user through a series of prompts, menus, or cards that results in a valid value for the chosen field.

OK

Accepts and saves the defined input data.

NEXT

Forwards the user to the next input data window for this entity.

PREVIOUS

Forwards the user to the previous input window for this entity.

CANCEL

Exits without saving or defining the data.

ENTER

Pressing ENTER at any time during input data definition accepts and saves the defined data.

2.16 LOCAL COORDINATE SYSTEM

VPG refers to the local coordinate system to translate, rotate, mirror, copy, and generate points, lines, and nodes. When such a function is selected, the program will automatically prompt the user to generate a local system designated as the UVW coordinate frame. One, two, or three reference points are required to establish a local coordinate system.

For a three-point system, the first reference point defines the local origin. The second reference, which extends from the first reference point, defines the direction of the local U-axis. The third reference point defines the local UV plane. The local W-axis is defined in the UV plane and is perpendicular to the U-axis. The local W-axis is then defined according to the right hand rule perpendicular to the UV plane.

For the two-point option, DONE should be selected after the second reference point is defined. The local W-axis lies along the vector from the origin to the second reference point, the V-axis lies in the VW plane, and the U-axis is defined by the right hand rule.

Note: All rotational commands (generating arcs, copying with rotation, etc.) are executed about the local W- or global Z-axis.

For the one reference point option, the user selects a point or node on the screen as the local origin then enters one of the X, Y, or Z options that are listed to define the local W along one of the global axes.

1. When creating a local coordinate system, VPG prompts:
   > CREATE LOCAL COORDINATES
   PICK NODE/PT FOR ORIGIN
   • VPG prompts for a desired coordinate system (local or global) in the options area.
   ? ABORT
   GLOBAL SYSTEM
   KEY IN X, Y, Z
   • This signals the user to key in the origin of the local coordinate system.
   > ENTER X, Y, Z COORDINATES

   • The user enters the values for the X, Y, and Z coordinates, e.g., 100,0, and 0.
   NODE
   • Default
   POINT
   SHOW LAST C.S.
   • Once the last coordinate system has been displayed, see step 3.
   • The user may select a node or a point, enter a coordinate by keyboard, or use any combination of these options to create a local coordinate system.
2. Once a reference point has been selected, VPG prompts:
   > PICK NEXT POINT OR NODE
   ? ABORT
   DONE
   • After the user selects 2 reference points, VPG displays the coordinate system and continues to step 3.
   KEY IN XYZ
   • The user may enter up to 3 reference points globally to define a local coordinate system.
   INCREMENTAL XYZ
   • The user may enter DX, DY, and DZ from the previous reference point to define a local coordinate system.
   NODE
   • The user may select up to 3 nodes to define a coordinate system.
   POINT

- The user may select up to 3 points to define a coordinate system.

REJECT LAST

- This allows the user to deselect the last reference point during the selection process.

X AS LOCAL W AXIS

- This defines a local coordinate system that is parallel to the global axis with the local W along the global X-axis and the local origin at the first reference point.

Y AS LOCAL W AXIS

- This defines a local coordinate system that is parallel to the global axis with the local W along the global Y-axis and the local origin at the first reference point.

Z AS LOCAL W AXIS

- This defines a local coordinate system parallel to the global axis with the local W along the global Z-axis and the local origin at the first reference point.

3. Once the user defines the desired coordinate system, VPG displays it and prompts:

> ACCEPT? (Y/N/A)

? YES

- VPG prompts for the next command.

NO

- The user returns to step 1.

ABORT

- The user returns to the menu.

2.17 ENTITY SELECTION

In certain commands such as COPY, DELETE, etc., VPG prompts the user to select elements, nodes, lines, surfaces, etc. A list of options will appear in the MENU WINDOW. The default selection option is cursor pick at the entity. Other commonly used selection options are described below:

WINDOW

A window (drag-window) is defined by clicking the left mouse button, dragging the cursor diagonally across the screen until the desired entities are within the window, and releasing the left mouse button to complete the selection. If an entity is partially outside the window, it will not be selected.

MULTI-PT REGION

A multi-point region (polygon) is defined by clicking the left mouse button in succession to enclose the desired polygon region. Click on the right mouse button to reject the last defined point. Click on the middle button to complete the region. Entities within this region will be selected. If part of an entity is outside of the region, it will not be selected.

PART

The part names will be listed in the menu area. Pick the name from the part list or pick an entity from the screen to select the part. Selected parts will be highlighted in white in the MENU WINDOW. All entities in the part will be selected.

REJECT LAST

This option negates the last selection whether from single cursor pick or a group of entities selected by any of the above options.

EXCLUDE ON/OFF

This option works like a toggle switch. If turned on, all the subsequent selected entities will be removed from the previously selected list. The user may toggle this option on and off during the course of the selection.

TYPE

This option is used to control the type of elements to be selected. If DONE is selected immediately after selecting TYPE, all elements of this type will be selected. Otherwise, other options may be used to limit the selection of elements.

Note: The TYPE function is specific to the selection of elements.

SURFACE

This function is used to select elements created from a surface by the SURFACE MESH command.

Note: The SURFACE function is specific to the selection of elements.

Chapter

3

Chapter 3: Getting Started

3.1 OPENING/CREATING AN eta/VPG DATABASE FILE

To start the VPG software, the user must execute the VPG program file, using the method appropriate for their operating system. In Windows this may be accomplished by double clicking the shortcut to the VPG executable file. Once the VPG software is activated, the VPG Open File window is displayed for the user to OPEN or CREATE a new VPG database.

1. See the Open File window below.

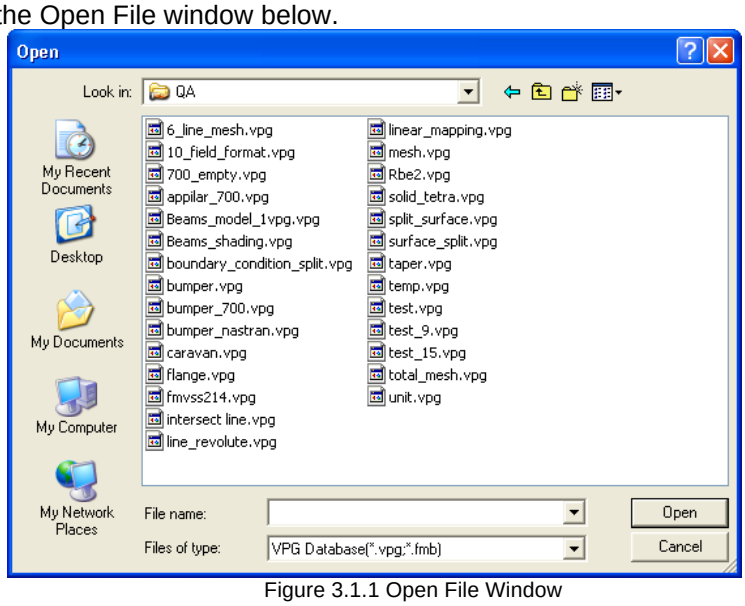


Figure 3.1.1 Open File Window

2.The user would either select the name of a previously saved file or enter the name of a new file in the Dialogue window. The recommended practice is to add the extension **.vpg** to a newly created file.

3.If creating a NEW file, the user would be prompted to do so:

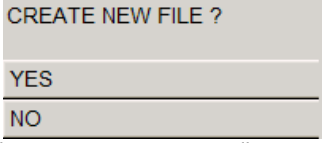


Figure 3.1.2 Create New File Prompt

4. The user will be prompted to select the analysis program desired:

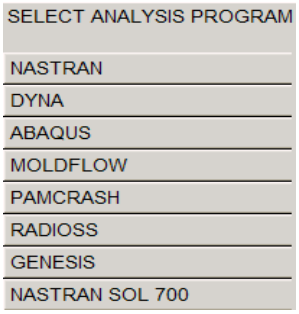
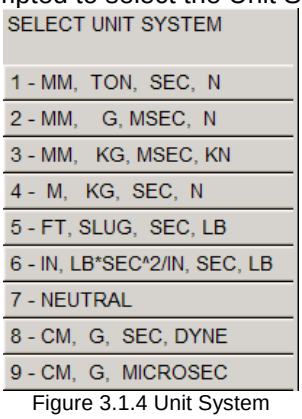


Figure 3.1.3 Select Analysis Type

The analysis program selected will set defaults for the eta/VPG session to generate either LS-DYNA or NASTRAN cards.

5. The user will also be prompted to select the Unit System desired:


SELECT UNIT SYSTEM
1 - MM, TON, SEC, N
2 - MM, G, MSEC, N
3 - MM, KG, MSEC, KN
4 - M, KG, SEC, N
5 - FT, SLUG, SEC, LB
6 - IN, LB*SEC^2/IN, SEC, LB
7 - NEUTRAL
8 - CM, G, SEC, DYNE
9 - CM, G, MICROSEC
Figure 3.1.4 Unit System

The selected unit system will be stored in the database as the default value setting. The user will now be in the MAIN MENU of eta/VPG and ready to start the session.

3.2 SETTING UP A VPG SYSTEM MODEL

Following is a general outline for setting up a VPG system model. There are three unique modules in the VPG pre-processor that allow the user to create a VPG model in an automated procedure. A detailed description of each of these processes is available in the VPG/Structure Tutorial.

1- Generating a Front Suspension Model

The user begins by selecting a front suspension model from the SUSPENSION MENU. The user determines the desired optional components such as stabilizers and steering system types and enters the node coordinates for suspension geometry.

Next, the user will be prompted to edit the default spring stiffness, damping coefficients, extra node coordinates, default mass, center of gravity, and inertia moment. Once the user has edited the default suspension values, the suspension will be displayed on screen.

Details on generation of suspension models and the types of suspensions available for generation through the VPG template system, may be found in Chapter 7, Section 2.

2- Generating a Rear Suspension Model

The rear suspension is generated in the same manner as a front suspension.

3- Generating Tire Models

After entering the TIRE MENU, the user selects GENERATE and is offered the TIRE PARAMETER dialogue box. Default values for tire geometry, mass and inflation pressure are edited to the user's parameters.

Once the tire geometry is defined, VPG prompts the user for the location of the tire. The user may define the location of the tire through an X, Y, Z position, or by selecting an existing node /point, or by selecting a location on an existing suspension system.

Tire modeling is discussed in Section 7.3. This topic is also described in the VPG/Structure Tutorial.

4- Auto-Generating a Road Surface

The user generates the desired road surface by selecting it from the ROAD SURFACE MENU. If the road surface is not in the correct position in relation to the suspension after generation, the MOVE ROAD SURFACE command is used to position it.

Next, contact between the road surface and the tires are determined using the FIVE_NODES_TO_SURFACE interface type. The road is then constrained in the Y, Z, and rotational directions using the material property assigned to the road. Next, a velocity is assigned to the road using the BOUNDARY_PRESCRIBED_MOTION card.

Details on ROAD SURFACE libraries may be found in Chapter 7, Section 1.

5- Adding a Body Model

The body to chassis/suspension attachment process depends upon what type of body model the user wishes to use for analysis (deformable or rigid).

Rigid Body - The rigid body model is constrained to the rigid beams that define the body attachment points on the suspension.

Deformable Body - The specific coordinates for the body attachment points must be entered when the user defines the Extra Node Coordinates for the front/rear suspension models. This ensures that the generated suspension would fit the specific body model. The user then creates weld spiders between the mounts on the vehicle and the rigid body beams on the suspension. Next the user defines the BODY_LOAD_DEFINITION_CARD to define the gravity.

After defining the gravity, the user pre-loads the suspension using the ELEMENT DEFINITION CARD.

6- Defining VPG Analysis Control Parameters

First, the CONTROL ENERGY and CONTROL TERMINATION cards must be defined.

Next, the output control interval data for the ASCII database must be determined. Then the BINARY DATABASE cards must be edited to control the output interval of the results and restarts.

7- Analysis Submission

From the ANALYSIS menu, select DYNA INPUT FILE OPTIONS, edit the dialogue box, and

submit the analysis. For additional information, please see Section 8.11.

8- Displaying Results

Enter the POST PROCESSING menu and select D3PLOT (LS-DYNA result file). Then select the analysis' 'd3plot' from the File Menu.

The results from each of the analysis steps will then be read into eta/VPG. A binary result file will be created at this time (named 'd3plot.pp'). Since the results are not saved to the eta/VPG database, this binary file should be reread into the post-processing menu when the user wishes to view the results again. This file is read much faster than the d3plot files.

The results are now ready to be post processed using a variety of features.

9- Graph Plotting

To graph the results, select GRAPH from the Post Processor Menu. The user then has the option of reading in a previously saved LS-DYNA ASCII graph file. When exiting the GRAPH MENU, the user has an option to save the graphs in a binary file for future processing. This file will be loaded much faster than the LS-DYNA ASCII files.

Chapter

4

Chapter 4: Main Menu

This is the first menu bar the user encounters when beginning a session with VPG. The initial options unfold into an additional series of submenus. These submenus are documented in the following sections.

File Parts Pre Road Suspension Tire Safety ALE-FSI DTM HEEDS Analysis Fatigue Post Utilities View Macro Help

Menus are selected by mouse pick and contain all eta/VPG functions. Each of these menus may be accessed through the Function Keys.

4.1 FILE

The FILE menu allows for the import and export of data into and out of the eta/VPG database. This data may be in the form of finite element data, CAD files, or a combination of the two.

The FILE menu also allows users to SAVE databases. It is important to know that eta/VPG does not automatically save the user's data. The user must execute the SAVE command to write the database changes to the file.

The FILE menu provides functions for RESTART, IMPORT and EXPORT functions.

4.2 PARTS

Models constructed in eta/VPG are organized into PARTS. These Parts may contain lines, surfaces, and/or elements. Parts may contain only one type of entity or may contain any combination of entity types. Parts may be created from the PART menu.

All data that is created in eta/VPG, such as elements, lines, or surfaces, is created in the CURRENT PART. The current part may be selected or changed from the PART menu. Once selected, all data created is placed in this part.

Parts may be turned on and off for display purposes from the PART menu.

4.3 PRE

The PRE menu is the main Preprocessing menu. In this menu, the user will find all of the necessary modeling functions to create elements, nodes, materials, contacts, element properties, and boundary conditions.

4.4 ROAD

The ROAD Menu allows users to select and place a road surface model in their VPG model. This

menu is only available if the user is in LS-DYNA mode (see UTILITIES menu, SETUP command).

4.5 SUSPENSION

The SUSPENSION menu allows users to create suspension system models from predefined templates. This menu is available only in LS-DYNA mode (see UTILITIES menu). Typical automotive suspension types are predefined, and users may input their data into the desired suspension template. VPG will then construct an LS-DYNA-based model using appropriate entities for the various model components.

4.6 TIRE

VPG contains tools to generate parametric tire models. These tire models are described in Chapter 7. This menu is available only in LS-DYNA mode (see UTILITIES menu).

4.7 SAFETY

This menu accesses the VPG/Safety Module. Access to this menu is available only in LS-DYNA mode (see UTILITIES menu).

4.8 ALE-FSI

VPG's ALE/FSI modules are used for fluid and solid coupling simulations.

4.9 DTM

This module allows users to set up a series of trop tests for a test specimen, using a few simple problem description templates.

4.10 HEEDS

HEEDS is an optimization program based on NASTRAN, DYNA etc, which will help to find a best result and do much good to the design. HEEDS module in VPG supports to define HEEDS discrete set, distribution, variable, tag and output HEEDS file and relative DYNA file. For tagging, it now only supports DYNA material and section

4.11 ANALYSIS

The ANALYSIS menu allows users to define all parameters necessary to execute an analysis. Many non-graphical parameters are required to carry out an analysis. For instance, LS-DYNA models require a termination time and require a definition of output frequency for the model results. VPG allows users to access all CONTROL CARDS and DATABASE parameters available in LS-DYNA. Similar capabilities are available for NASTRAN, RADIOSS and PAMCRASH model databases.

4.12 FATIGUE

The FATIGUE menu allows users to post process stress data calculate fatigue damage for LS-DYNA or NASTRAN data. This menu allows users to calculate and display the results of this fatigue calculation.

4.13 POST

The POST menu launches the ETA Postprocessor application. By selecting this menu option, the current database is stored in a temporary location and closed. The Post Processor application is then opened. Once the Postprocessor application is closed, the temporary database is then restored.

4.14 UTILITIES

The UTILITIES menu allows users to perform various modeling and display functions. As an example, the user may add Arrows or Comments on the screen for additional information when creating JPEG or GIF image files.

An important feature of the UTILITIES menu is the SETUP command. This command allows the user to specify display and model database characteristics. From this menu the user may specify the model type and global parameters to be applied to the model.

On the SETUP menu, the user may specify the analysis program for the current database. This function defines the menus that will be displayed, as well as the content of those menus. As an example, when the NASTRAN option is selected for this command, the menus will then contain the material types, element types, and boundary conditions that pertain to NASTRAN models. All analysis parameters will also be specified in a manner consistent with NASTRAN. If the user selects LS-DYNA, similarly, all menus, material models, contact types, and element types will be consistent with LS-DYNA models.

4.15 VIEW

This menu allows users to manipulate the model in the Display Window. Views may be saved and recalled by the user.

4.16 MACRO

The MACRO menu allows the user to record commands executed by VPG and the replays those commands in an automated fashion.

4.17 HELP

The Version submenu allows user to identify the build date and version of the VPG software. The Online Help menu provides access to the on-line version of this user manual, in a searchable format.

Chapter 5

Chapter 5: File Manager

The options available in FILE MENU allow the user to input data from different analysis programs into VPG. A detailed description of each function in the FILE menu is given in the following sections.

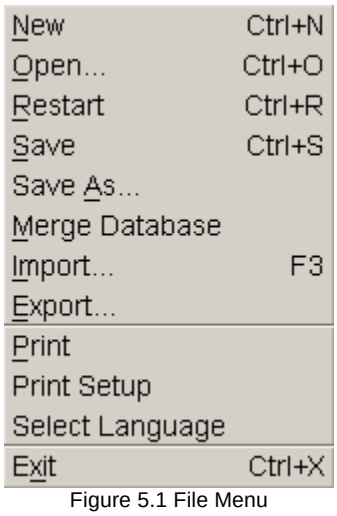


Figure 5.1 File Menu

5.1 NEW

This function allows the user to create a new database file.

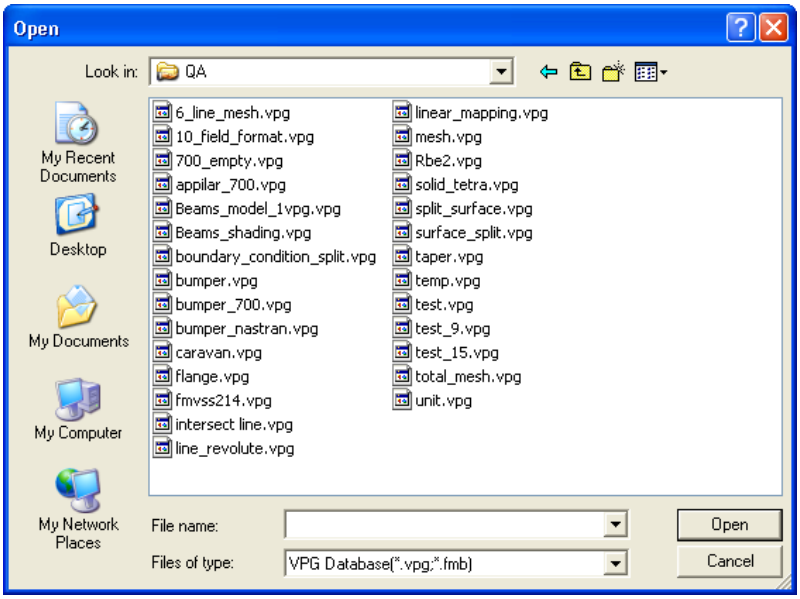


Figure 5.1.1 New File Window

If a database file already has been opened in VPG and has not been saved yet, there is a warning message for saving the opened file before creating a new one. See Figure 5.1.2.

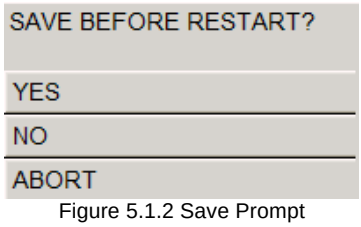


Figure 5.1.2 Save Prompt

5.2 OPEN

This function allows the user to open databases.

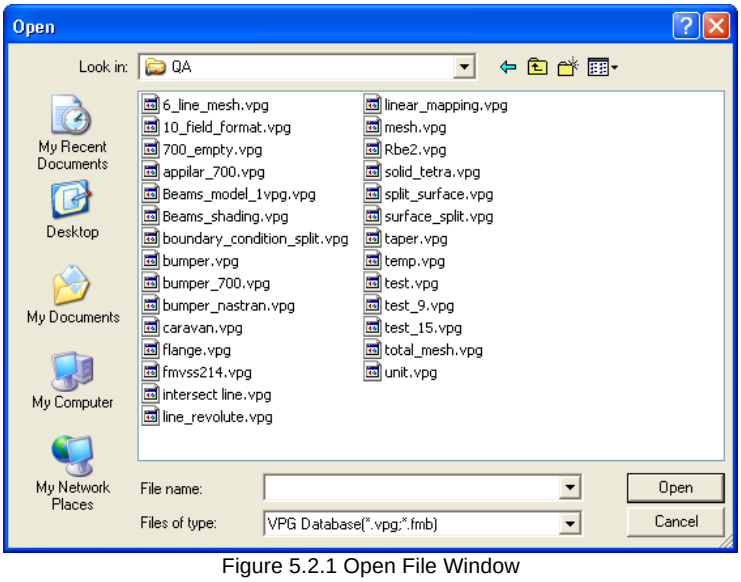


Figure 5.2.1 Open File Window

If a database file already has been opened in VPG and has not been saved yet, there is a warning message for saving the opened file before opening another one. See Figure 5.2.2.

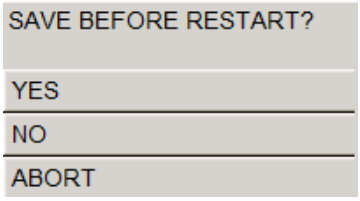


Figure 5.2.2 Save Prompt

5.3 RESTART

This function allows the user to restart the current database from the last saved point. VPG will prompt the user to save the current file. See Figure 5.2.2.

After that, VPG will prompt the user to restart from a new database or the current one.

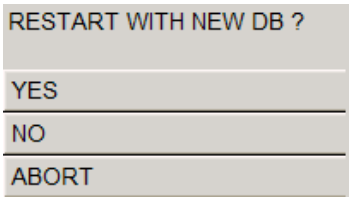


Figure 5.3.1 Restart Database

- Select YES to open a new database. The Open File Window appears.
- Select NO to reopen the current database from the last saved point
- ABORT to cancel the operation

5.4 SAVE

This function updates the current database. The User may overwrite the existing file or create a new file at the execution of the SAVE command. If the user chooses to overwrite the file, all previous mode data will be overwritten and the current model configuration will be saved. If the user chooses to create a new file, the current database will not be updated and the current model configuration will be written to a new file, with the database name provided by the user. This file now becomes the current model database open in VPG.

5.5 SAVE AS

This function saves the current database as a new file. See Figure 5.5.1

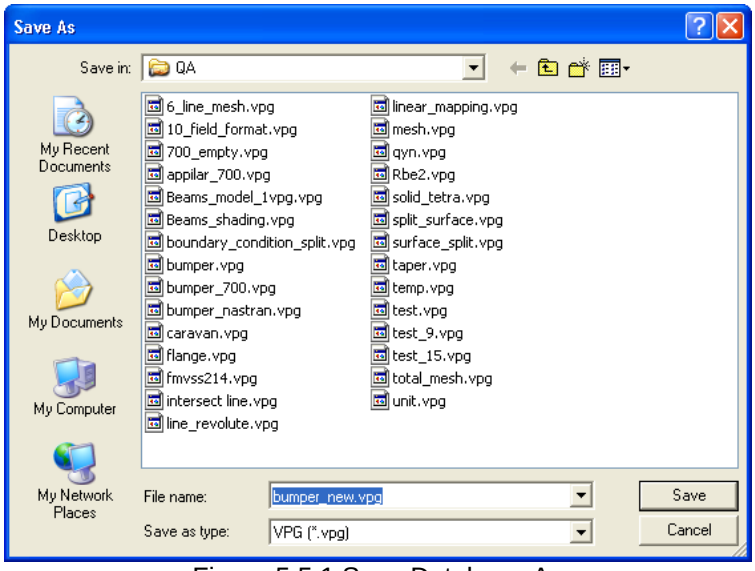


Figure 5.5.1 Save Database As

5.6 MERGE DATABASE

This function adds another VPG database into current database. See Figure 5.6.1

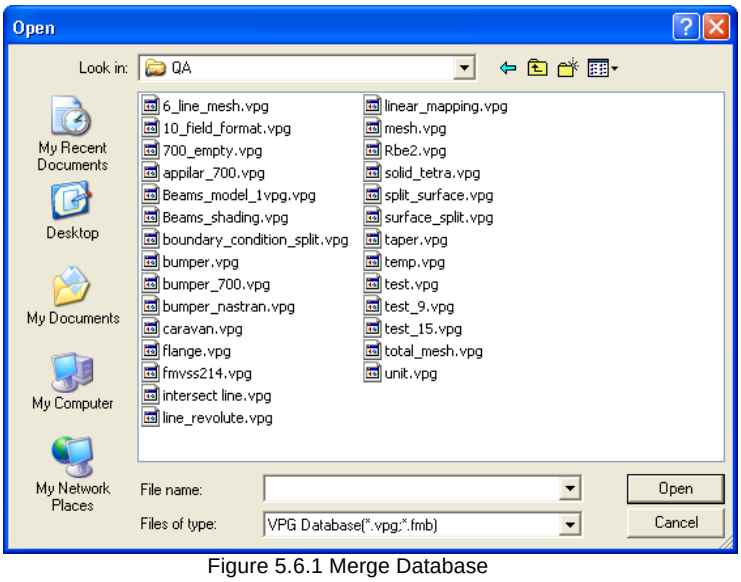


Figure 5.6.1 Merge Database

5.7 IMPORT

This function allows the user to read CAD or model data (See Figure 5.7.1). User can import more than one CAD or model data at a time by using CTRL or SHIFT key.

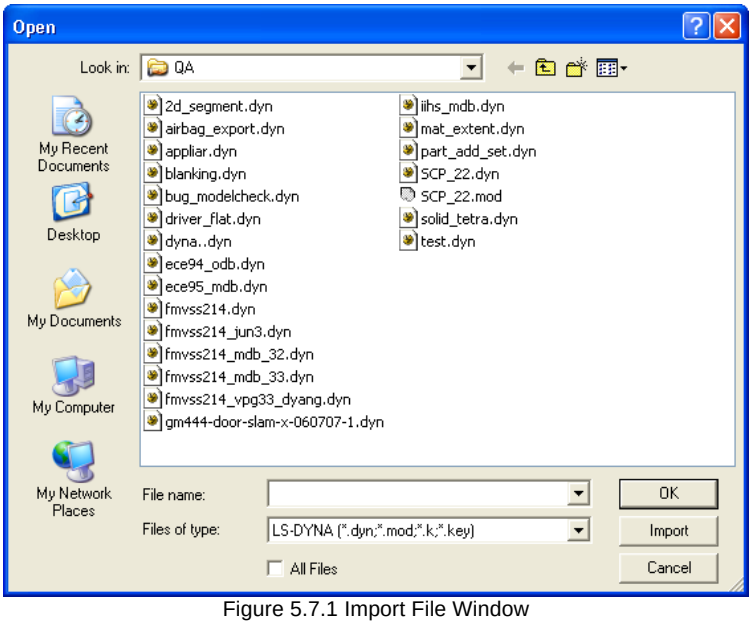


Figure 5.7.1 Import File Window

5.7.1 FILE TYPE

Select the drop down selection in the FILE TYPE field to select the proper format. The available formats are:

MODEL DATA FORMAT: LS-DYNA, Nastran, Abaqus, Ansys, Genesis, Moldflow, PamCrash, Radioss, Nastran SOL 700
CAD DATA FILE: IGES, LINE, VDA, DXF, CATIA, STEP, Unigraphics, Proe, Parasolid, Solidworks, Ideas.
SPECIAL MODEL FORMAT: ADAMS, ADAMS-Pre

A detailed description of each file import function is given in the following section.

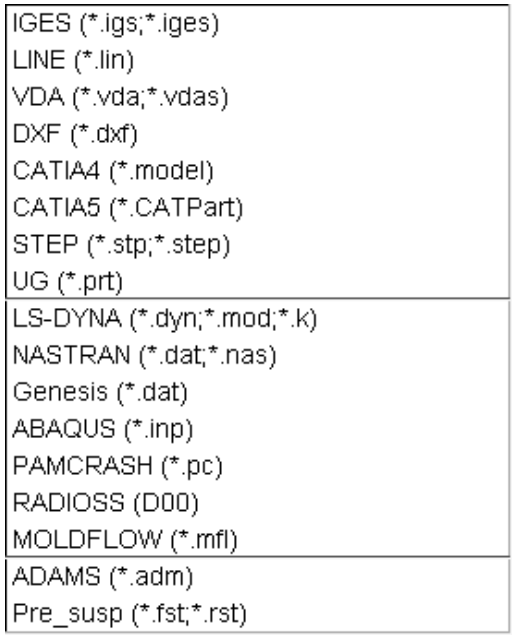


Figure 5.7.2 File Format

5.7.1.1 READ CAD DATA

A series of functions in this menu allow the user to read converted wireframe and surface data into VPG. The supported file formats include:
IGES (*.igs, *.iges), VPG (*.lin), VDA (*.vda), DXF (*.dxf), CATIA4 (*.model), CATIA5 (*.catpart), STEP (*stp,*.step), and UG (*.prt)

In order to provide flexibility in accepting line data from different CAD systems, VPG uses a neutral line data format to communicate with these CAD systems: IGES, PDGS standard, CGS (INCA and DES), DXF, etc. Once the user has entered the command, the files with the appropriate suffix will be listed in the options area. The user can then select the desired file.

5.7.1.2 READ ABAQUS FILE

VPG reads ABAQUS files directly with its built in translator. The VPG ABAQUS file extension is .**inp.**

5.7.1.3 READ DYNA FILE

This function allows the user to read both keyword and non-keyword LS-DYNA (.**dyn**) files directly into VPG. VPG supports versions 88 to 971 Keyword. Once the user has entered the command, the files with the suffix **.dyn** will be listed in the VPG FILE WINDOW. The user can then select the desired file.

5.7.1.4 READ GENESIS FILE

This function allows users to read GENESIS (.**dat)** files directly into VPG. Once the user has entered the command, the files with the suffix **.dat** will be listed in the options area. The user can then select the desired file.

5.7.1.5 READ MOLDFLOW FILE

This function allows users to read MOLDFLOW data directly into VPG.

1. VPG prompts:
   > **READING MOLDFLOW INPUT FILE**
   > **ENTER THE ROOT NAME OF THE MOLDFLOW INPUT FILES**

2. After entering the root file name (assuming that the **.mfl, .mod, and .tri** files are available), VPG displays the MOLDFLOW model.

3. If the user is reading data into an existing model, VPG prompts:

   > **OFFSET NODE AND ELEMENT NUMBERS. (Y/N)**
   - **YES** renumbers the new data at the lowest available unused node/element number.
   - **NO** compares the node/element numbers and ignores the duplicates -- VPG retains only the original nodes and elements (i.e., if the MOLDFLOW file contains all duplicate node and element numbers, the new part will read in with out the presence of elements or nodes).

4. VPG reads in the file and returns the user to the **FILE MANAGER** menu.

5.7.1.6 READ NASTRAN BULK

VPG uses NASTRAN as a file translator so that the user may import and export models and mesh. This function allows the user to read a NASTRAN bulk data file (**.dat**) directly into VPG. All existing properties, materials, and subcases are retained.

1. VPG displays the VPG FILE MENU and prompts:
   > DEFINE NASTRAN BULK DATA FILE

2. VPG prompts:
   > SELECT PART CONTROL OPTION FOR CBARS
   The files with the suffix .nas will be listed for you in the options area.

MAT1 ID
- CBARS will be grouped by common MAT1 ID.

PBAR ID
- CBARS will be grouped by common PBAR ID.

PART NAME
- CBARS will be grouped by their individual part names.

3. If the user is reading data into an existing model, VPG prompts:
   > OFFSET NODE AND ELEMENT NUMBERS. (Y/N)
   - YES renumbers the new data at the lowest available unused node/element number.
   - NO compares the node/element numbers and ignores the duplicates—VPG retains the only original nodes and elements (i.e., if the NASTRAN file contains all duplicate node and element numbers, the new part will read in without the presence of elements or nodes).

4. VPG reads in the file and returns the user to the FILE MANAGER menu.

5.7.1.7 READ PAMCRASH FILE

This function allows the user to read a PAMCRASH (**.pc**) file directly into VPG. Once the user has entered the command, the files with the suffix **.pc** will be listed in the options area. The user may then select the desired file.

5.7.1.8 READ RADIOSS FILE

This function allows the user to read a RADIOSS data file directly into VPG. VPG can read fixed format input files from RADIOSS Versions 2.1, 2.2, 2.3, 3.1, 4.1and 4.4. VPG can write RADIOSS 4.4 fixed format input files. VPG supports all input cards of RADIOSS 4.4 fixed format. VPG contains options when reading RADIOSS input and output files. These options relate to the translation of rigid bodies to VPG and the loading of model information. RADIOSS requires two files to be loaded- one is the RADIOSS output file (.out), which contains the rigid body information. The second is the RADIOSS input file (.D00), which contains model information. If the user has both files, he should click YES at the first prompt to read both the RADIOSS output and input files. If the user has only the RADIOSS input file, the user should select NO.

1. Begin with reading the RADIOSS output file (.out). This loads the rigid body information. VPG will prompt:
   > ADJUST RIGID BODY PRIMARY NODES FROM RADIOSS OUTPUT. (Y/N)
   - If **YES** is selected, the RIGID BODY information will be loaded with the output file. The user can then load the RADIOSS input file (.D00).
   - If **NO** is selected, VPG will prompt:

   > ADJUST RIGID BODY PRIMARY NODES BY VPG. (Y/N)

- **YES** will locate the RIGID bodies and recalculate the rigid body primary node locations.
- **NO** will finish the input sequence without modifying the model information.

VPG creates some default materials and element properties for elements that are lost during the file reading. These are assigned to the part DEFAU_#. VPG creates a node set, element set, material set, property set, and interface (contact) set for boundary conditions, interface, rigid walls, time history etc. VPG can read multiple RADIOSS input files into the same database.

5.7.1.9 READ NASTRAN SOL 700 FILE

This function allows the user to read a NASTRAN SOL 700 (**.dat**) file directly into VPG. Once the user has entered the command, the files with the suffix **.dat** will be listed in the options area. The user may then select the desired file.

5.7.1.10 READ ANSYS FILE

This function allows the user to read a file of ANSYS format *.cdb or *.anf. VPG supports to read in and write out Element, Node and Part information of FE model.

5.7.1.11 READ SUSP (ADAMS)

Section J of the Appendix gives a complete description of converting ADAMS models to VPG. Please refer to that section when converting an ADAMS suspension.

5.8 EXPORT

This function allows the user to output from the current database. The options are similar to the options mentioned above. See Figure 5.8.1.

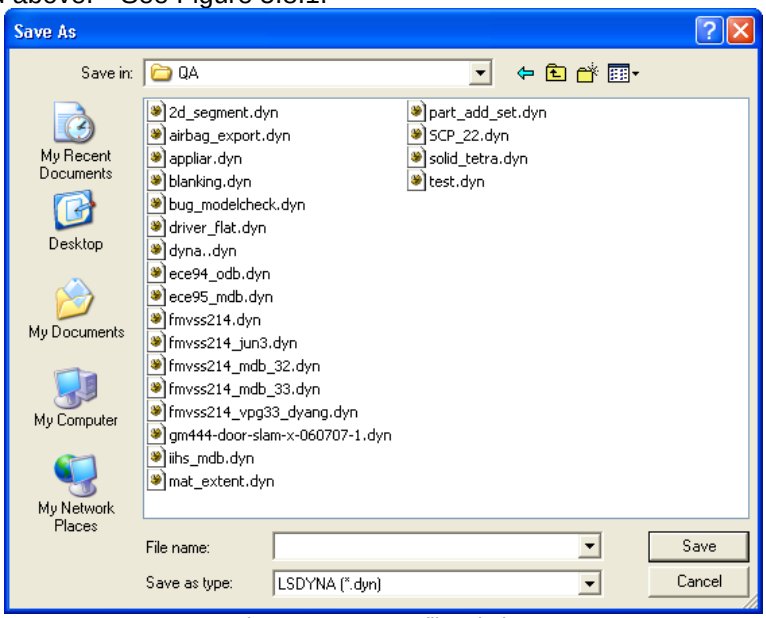


Figure 5.8.1 Export file Window

A detailed description of each file export function is given in the following section.

5.8.1 WRITE ABAQUS FILE

VPG displays the CONTROL PARAMETER window. Users may select the parameters with the mouse button and enter new values through the keyboard. (The parameters are ABAQUS specific. Refer to the ABAQUS manual for more details.)

1. Enter a file name (up to 24 characters). Using the **.inp** convention for ABAQUS files is suggested.

2. If the file name already exists, VPG prompts the following message:
   >FILENAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS. (Y/N)
   - **YES** will only output the active parts (parts that are turned ON).
   - **NO** will output all parts in the database.

4. VPG prompts:
   > INPUT THE HEADING OF THE FILE
   - The user can enter up to 60 characters.

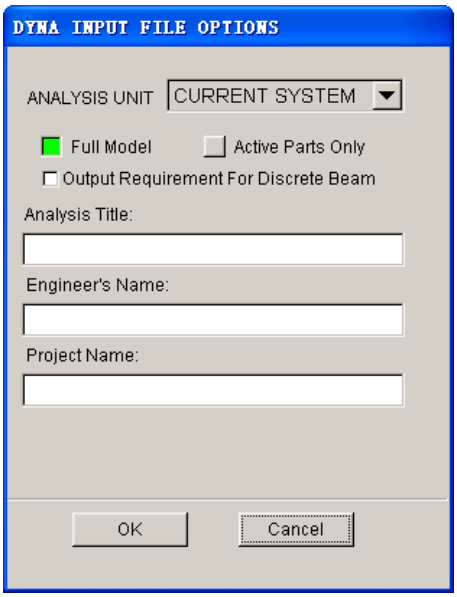
5.8.2 WRITE DYNA FILE

This function allows the user to output an LS-DYNA data deck directly from the VPG database.

1. Enter a file name (up to 24 characters). The **.dyn** extension is suggested for LS-DYNA files.

2. If the file name already exists, the program will prompt the following message:
   > FILE NAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)
   - YES will overwrite the existing file.
   - NO will prompt for a file name.

3. User selects ANALSYS UNIT, parts output mode (FULL MODEL or Active Parts Only), discrete beam output option, inputs Analysis Title, Engineering's Name and Project Name.



4. After clicking OK, VPG displays the CONTROL TERMINATION CARD if a termination time has not been specified yet. Once the card is defined and the user selects OK, the DYNA deck is written.

5.8.3 WRITE GENESIS FILE

This function allows the user to output a GENESIS (.dat) data deck directly from the VPG database.

5.8.4 WRITE LINE DATA FILE

This function allows the user to output a LINE/SURFACE DATA (.lin) data file directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters). The **.lin** extension is suggested.

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)
   ? YES
   NO
   - YES will overwrite the existing file.
   - NO will prompt for a file name.

3. VPG prompts:
   > WRITE LINES/SURFACES ONLY IN ACTIVE PARTS. (Y/N)
   ? YES
   NO
   - YES will only output the lines/surfaces displayed on the screen.
   - NO will output all elements' lines/surfaces in the database.

5.8.5 WRITE MOLDFLOW FILE

This function allows the user to output the current model data into an external file in MOLDFLOW format. VPG supports Moldflow version 5.1.

Note: Node and element numbers must be in sequence to write a MOLDFLOW file. The user may either renumber or compress nodes/elements (see NODE OPTIONS menu). If the nodes are not in sequential order, VPG prompts:

> **RENUMBER NODE NUMBERS AND TRY AGAIN (or)**
> **RENUMBER ELEMENT NUMBERS AND TRY AGAIN**

1. VPG prompts:
   > **ENTER ROOT NAME FOR INPUT FILES OR STOP TO EXIT**
   - VPG writes three files in accordance with the MOLDFLOW format (**.mfl, .mod, and .tri).**

2. If the file name already exists, the program will prompt the following message:
   >FILENAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

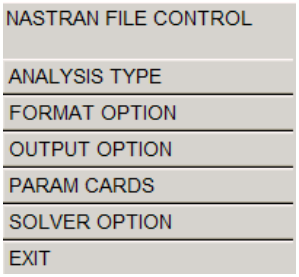
3. After entering file name, VPG prompts:
   > **SELECT UNIT CURRENTLY USED IN DATABASE**
   ? **MM**
   CM
   IN

4. VPG prompts:
   > **OUTPUT ONLY ACTIVE PARTS. (Y/N)**
   - **YES** will only output the elements displayed on the screen.
   - **NO** will output all elements existing in the database.

5. VPG prompts:
   > **ENTER TITLE**

5.8.6 WRITE NASTRAN

This function allows the user to output the current model data to an external file into NASTRAN format. The user may output either the complete model data or partial model data. For detailed descriptions of these options, users should refer to NASTRAN software documentation.



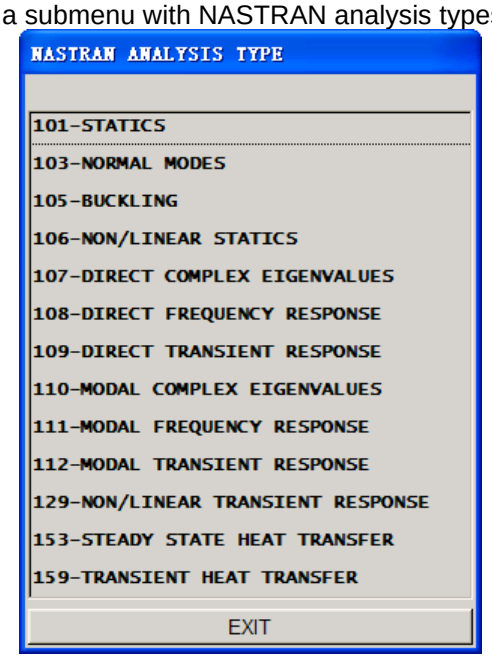
Detailed description:

1. VPG Prompts:

> ENTER CONTROLS OPTION

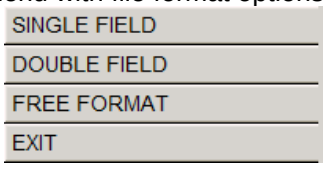
ANALYSIS TYPE

- Displays a submenu with NASTRAN analysis types:



FORMAT OPTION

- Displays a submenu with file format options:



SINGLE FIELD (fixed format columns of eight)
DOUBLE FIELD (double precision)

OUTPUT OPTION

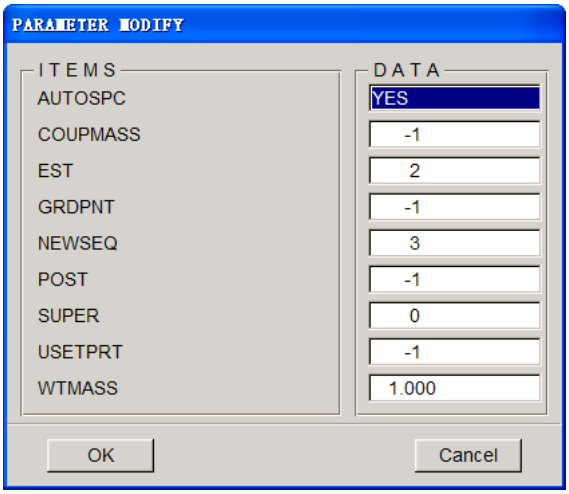
- Displays a submenu with output options:

PARAM CARDS

- Displays the parameter modify window (see figure below).

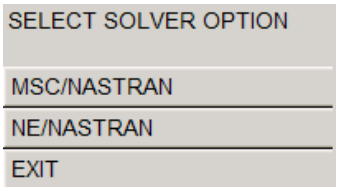
WRITE FILE

EXIT



SOLVER OPTION

- Prompts user to select NASTRAN solver:



2. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters).

3. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS. (Y/N)
   ? YES
   NO
   - YES will only output the elements displayed on the screen.
   - NO will output all elements existing in the database.

4. VPG prompts:
   > ENTER ANALYSIS TITLE
   - Any alphanumeric input up to 72 characters may be entered as the title of the NASTRAN bulk data file.

5.8.7 WRITE PAMCRASH FILE

This function allows the user to output a PAMCRASH (.pc) data file directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters).

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   - **> SELECT PAMCRASH VERSION:**
   - **? 1 - SEQUENTIAL**
     2 - FREE NUMBERING
   - **> DO YOU WANT TO USE KEYWORD INPUT (Y/N)**
   - **? YES / NO**

4. VPG prompts:
   - **> OUTPUT ONLY ACTIVE PARTS. (Y/N)**
   - **? YES / NO**

5. VPG prompts for optional keyword cards via pop ups.

6. VPG prompts:
   - **> ENTER TITLE**
   - The user enters an alphanumeric title containing a maximum of 48 characters.
   - VPG returns the user to the FILE MANAGER menu.

Note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the properties/materials by default, by global default, or by part. Supported properties are listed in Appendix A, VPG Capabilities for F.E.A. Programs.

5.8.8 WRITE RADIOSS FILE

This function allows the user to output a RADIOSS data file directly from the VPG database. When writing a RADIOSS input file, VPG creates default materials and element properties for some elements. For specific translation properties, see the RADIOSS conversion tables.

1. VPG prompts:
   - **> ENTER FILE NAME OR "STOP" TO EXIT**
   - Enter a file name (up to 24 characters).

2. If the file name already exists, the program will prompt the following message:
   - **> FILENAME ALREADY EXISTS, O.K. TO OVERWRITE. (Y/N)**
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   - **> SELECT RADIOSS VERSION**
   - **? 1 - VERSION 4.1 (FIXED)**
     2 - VERSION 4.4 (BLOCK)

4. VPG prompts:
   - **> ENTER TITLE**
   - The user enters an alphanumeric title containing a maximum of 48 characters.
   - **VPG** returns the user to the **FILE MANAGER** menu.

5. VPG prompts:
   - **> OUTPUT ONLY ACTIVE PARTS. (Y/N)**
   - **YES** outputs only the elements displayed on the screen.
   - **NO** outputs all of the existing elements in the database.
   - After entering the appropriate selection, the user returns to the FILE MANAGER menu.

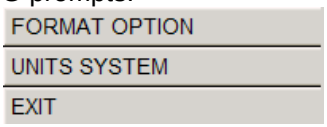
Note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the

properties/materials by default, by global default, or by part. Supported properties are listed in Appendix A, VPG Capabilities for F.E.A. Programs.

5.8.9 WRITE NASTRAN SOL 700 FILE

This function allows the user to output a NASTRAN SOL 700 (.dat) data file directly from the VPG database.

1. VPG prompts:



- **FORMAT OPTION**
  User selects output format as single field or double field.
- **UNIT SYSTEM**
  User selects unit system for output file.

2. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS. (Y/N)
   ? YES / NO

3. VPG prompts to enter termination time. The output file is written after entering this value.

5.8.10 WRITE ANSYS FILE

This function allows the user to output a ANSYS (.cdb) data file directly from the VPG database.

5.9 PRINT

This function creates a postscript file of display area and sends the file to the printer (default) or to a file. Prior to printing, the postscript driver must be initialized to accommodate the eta/VPG software. The user can define the printing default.

5.10 PRINT SETUP

This function allows the user to define the default setting for printing of image files. The options are shown in Figure 5.11.1.

5.10.1 PAPER CUT

The user can specify a given paper size and margin. Standard paper sizes are supported, such as 8.5”x11”, 11”x17”, A4, etc.

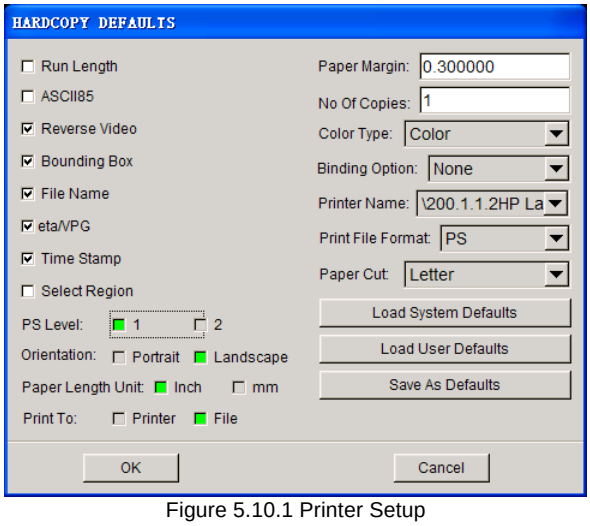


Figure 5.10.1 Printer Setup

- Select drop list at the right of **PAPER CUT** to choose a paper size.
- Enter a number in the field beside **PAPER MARGIN** to define the shorter of the paper's two margins (top and bottom).
- Select the button beside **Paper Length Unit** to choose a unit (in or mm).

5.10.2 PRINTER

- The user can select a printer or select a format to print to a file
- Enter the printer name in field or select from the list below
- If **Print To File** is toggled on, the user can choose a format by selecting the drop list beside **Print File Format**. VPG supports postscript (PS), encapsulated postscript (EPS), .GIF, and .JPG **file formats.**

Note: When PRINT is selected, the user will be prompted enter a file name instead of the File being directly sent to the printer

5.10.3 ORIENTATION

This function sets the page orientation as landscape or portrait.

5.10.4 COLOR TYPE

Select the drop list beside Color Type to select a color type.

5.10.5 POSTSCRIPT

There are two levels. Level 2 is the default setting. Level 1 should be selected when using an older model PostScript printer.

5.10.6 RUN LENGTH ENCODE

This option reduces the Postscript file size. Usually the reductions are dramatic (for PS Level 1, this is automatically turned off).

5.10.7 ASCII85 ENCODE

This option reduces the Postscript file size. Usually the reductions are dramatic (for PS Level 1, this is automatically turned off).

5.10.8 BINDING OPTION

This function leaves binding space along the top or left margins of the print and selects non-Binding space.

5.10.9 REVERSE VIDEO

This function reverses the black and white colors image in some cases. This feature affects only the background of the image.

5.10.10 BOUNDING BOX

This function draws a line frame around picture's border.

5.10.11 TIME STAMP

This function prints the current time at the lower right corner of the picture file.

5.10.12 FILE NAME

This function prints the file name at the lower right corner of the picture.

5.10.13 eta/VPG

This function prints eta/VPG logo at the lower left corner of the picture.

5.10.14 SELECT REGION

This function allows the user to define a graphic region by using a drawing window. Only the graphics within the region will be written to the graphics file. If the option is off (default), the complete graphics region will be saved to the output file.

Note: The user will be prompted to define the region (drag window) after PRINT is selected

5.10.15 NO.OF COPIES

This function allows the user to print multiple copies.

5.10.16 SETUP SAVE/LOAD

Default page settings may be used, saved, and selected by the user. All parameters are defined in the HARDCOPYDEFAULT file in the VPG installation directory.

5.10.16.1 LOAD SYSTEM DEFAULT

This function will load the system defaults that are saved in a VPG Hardcopy Default file located in the VPG executable directory.

5.10.16.2 LOAD USER DEFAULT

This function will load the user-defined defaults that have been saved in a HardcopyDefault file located in the user's home directory.

5.10.16.3 SAVE AS DEFAULT

This function saves user-modified hardcopy options to a HardcopyDefault file located in the user's Home directory.

5.11 EXIT

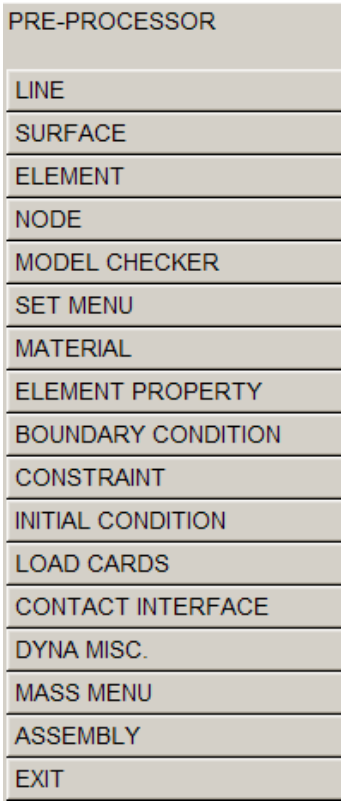
This function allows the user to exit the program. VPG will prompt to save current database.

Chapter

6

Chapter 6: PRE PROCESSOR

The Pre Processor menu contains a series of submenus with the ability to create and edit lines, surfaces, elements, material properties, and element properties.

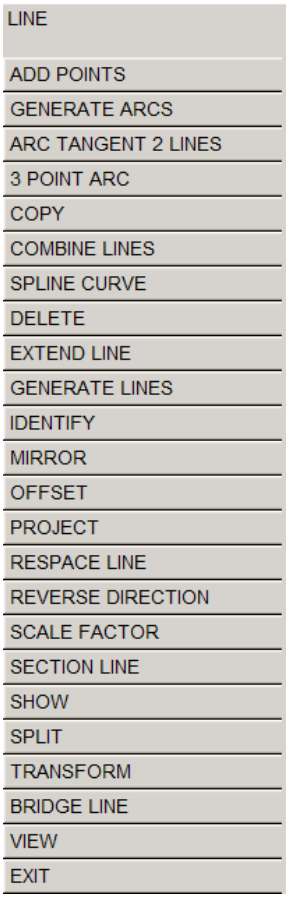


The Pre Processor submenus are described in their own individual sections that follow.

Note: DYNA MISC only appears when the analysis code is set to LS-DYNA. SUPERELEMENT only appears when the analysis code is set to NASTRAN.

6.1 LINE

The functions of the **LINE MENU** are intended to create a new set of line data or to modify existing line data. VPG currently has a limitation of 150,000 lines or 200,000 points per database. The following options are available in the **LINE MENU**:



Each option is selectable by using the left mouse button to select the menu item or by typing the first two letters of each command in the prompt in the VPG Message Window. A detailed description of each function is given in the following sections.

6.1.1 ADD POINTS

Description: This function allows the user to add points to an existing line either randomly or between two selected points.

Usage:

1. VPG prompts:
   > SELECT LINE FOR ADDED POINTS

2. VPG prompts:
   > SELECT POINT LOCATION ON LINE
   - The user may create new points by placing the cursor anywhere on the line and pressing the left mouse button, or the user may select one the following options:

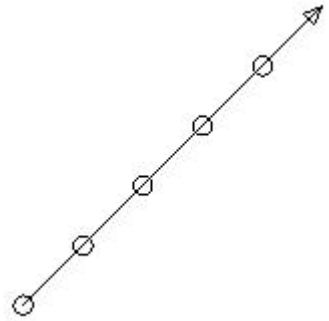
   ? BETWEEN 2 POINTS

- This option allows the user to select 2 end points of a line segment and create a specified number of new points between them (see step 3).

CURSOR LOCATION
POINT
REJECT LAST

3. VPG prompts:
   > BETWEEN 2 POINTS
   > SELECT POINT 1 ON LINE
   > SELECT POINT 2 ON LINE
   > ENTER NUMBER OF POINTS TO BE ADDED
   - The user enters the desired number of points.

4. EXIT or DONE returns the user to the LINE menu.



Circles indicate the locations where points have been added.

6.1.2 GENERATE ARCS

Description: This function allows the user to generate arcs and circles either globally, locally, or about a selected node or point. The created arc will be included in the CURRENT PART (see PART CONTROL MENU).

Usage:

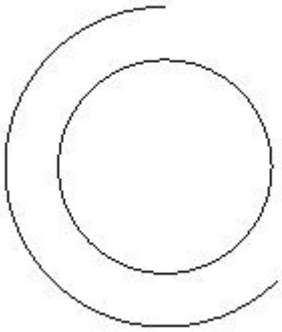
1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global) about which to create the arc. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in the Local Coordinate System, Section 2.16.

Note: Arcs and circles are generated about the local W-axis or global Z-axis.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)

3. If accepted, VPG prompts:
   > ENTER RAD, TH1, TH2, INC
   - RAD = radius of arc.
   - TH1 = beginning point of arc or circle.

- TH2 = ending point of arc or circle.
- INC = angle increment between points (default is 5 degrees).
- Any real value greater than zero is valid. i.e., 100,0,360,10 is a circle with a radius of 100 units with an increment of 10 degrees between points.



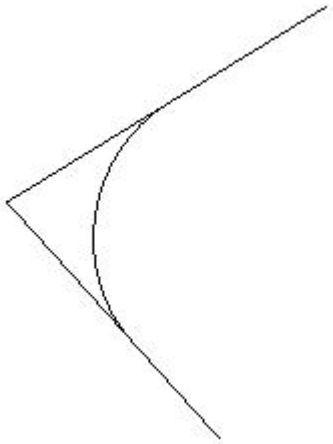
An arc and a circle have been generated with the same center.

4. VPG prompts:
   > SAME CENTER AND PLANE? (YES/NO/EXIT)
   - Do you want to create another circle/arc using the same center and plane.
   - **YES** returns the user to step 3.
     NO returns the user to step 1.
     EXIT exits the function.

6.1.3 ARC TANGENT 2 LINES

Description: This command allows the user to create an arc tangent to 2 lines. The arc will be included in the CURRENT PART (see PART CONTROL MENU). The tangent arc will trim the two intersecting lines.

Usage:



Arc tangent to two lines.

1. VPG prompts for a line:
   > SELECT LINE 1

2. VPG prompts for a second line:
   > SELECT INTERSECTING LINE

3. VPG prompts:
   > ENTER RADIUS OF ARC OR -1 TO EXIT
   - Any real value greater than zero is valid.
   - An arc tangent to the 2 lines is drawn. The lines need to be long enough to make actual contact with the arc but do not need to intersect.

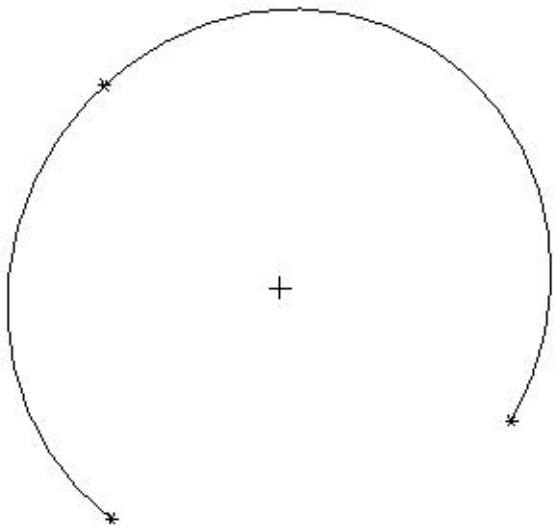
6.1.4 3 POINT ARC

Description: This function allows the user to create an arc through 3 points/nodes that are not collinear. The created arc will be included in the current part.

Usage:

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   - This command will be prompted 3 times.
   - When 3 points or nodes are selected an arc will automatically be drawn.

2. EXIT will terminate this function.



The asterisks represent the points through which the arc is drawn.

6.1.5 COPY

Description: This function allows the user to generate a set of duplicate lines. The user may then simultaneously translate and/or rotate the copied lines.

Usage:

1. VPG prompts:
   > PICK LINE TO COPY
   > SELECT LINES

- Select **DONE** once the desired lines have been selected.

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

   - Once the desired coordinate system is acquired it will be displayed on the screen.

3. VPG prompts:
   > ACCEPT? (Y/N/A)
   ? YES / NO
   - If **YES**, then go to step 4.

   - If **NO**, then go to step 2.

   ABORT
   - If **ABORT** is selected, then the function is terminated.

4. VPG prompts:
   > ENTER NUMBER OF COPIES (positive integer)

5. VPG prompts:
   > MOVE OR ROTATE (M/R)
   ? MOVE / ROTATE/ ABORT

   - If **MOVE** is entered,
     > ENTER U, V, W INCREMENTS
     - i.e., 0,0,1000 will copy the selected line(s) 1000 units along the W-axis.

     > INCLUDE COPIED LINE IN ITS ORIGINAL PART (Y/N)
     - Allow the user to specify the desired part.

   - If **ROTATE** is entered,
     > ENTER ANGLE INCREMENT (degrees)
     - i.e., 200 degrees.

     > INCLUDE COPIED LINE IN ITS ORIGINAL PART (Y/N)
     - Allow the user to specify the desired part.

   - If **ABORT** is entered, then the function is terminated.

6.1.6 COMBINE LINES

Description: This function allows the user to combine multiple lines into a new single line with a new line number. The original lines will be deleted after they are combined. The combined line should be selected in a logical sequence to form the new line.

Usage:

1. VPG prompts:
   > PICK LINES TO COMBINE

2. Once all the lines are selected, **DONE** or **EXIT** combines the lines and exits the command.

6.1.7 SPLINE CURVE

Description: This function allows the user to draw a spline curve line through multiple points or nodes or any combination of points or nodes. A minimum of 3 Points or Nodes is required for use of this function.

Usage:

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   ? ABORT
   DONE
   EXIT
   KEY IN XYZ
   - The user may enter up to 3 points globally to define a point or node.

   INCREMENTAL XYZ
   - The user may enter DX, DY, DZ from the previous point selected to define a point or node.

   NODE
   POINT
   REJECT LAST

2. **DONE** and **EXIT** will exit the function.

an example of a generated spline curve.

6.1.8 DELETE

Description: This function allows the user to delete selected lines. Lines may be selected by individual cursor selections, by dragging a window, by entering line numbers or by multi-point region selections.

Usage:

1. VPG prompts:
   > PICK LINE TO DELETE
   > SELECT LINES
   - Select **DONE** once the desired lines have been selected.

6.1.9 EXTEND LINE

Description: This function allows the user to extend a line by a specified distance along the axis of the line at either end of the line.

Usage:

1. VPG prompts:
   > SELECT LINE TO EXTEND

2. VPG prompts:
   > SELECT END POINT TO EXTEND LINE
   • Select the desired end point of the line to extend.

3. VPG prompts:
   > ENTER LENGTH TO BE EXTENDED
   • Input any real number.

4. Selecting **UNDO** will undo the last extend operation.

6.1.10 GENERATE LINES

Description: This function allows the user to draw a line between nodes or points via a combination of keyboard and mouse selections. The created lines are incorporated into the CURRENT PART (or see PART CONTROL MENU).

Usage:

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   ? INCREMENTAL XYZ
   • The user may enter an additional DX, DY, DZ from the previous point or node selected to continue drawing the line.
   KEY IN XYZ
   • The user may enter up to 3 points globally to define a point or node.
   NODE
   POINT
   WORK PLANE
   • Select nodes from work plane to generate lines
   WORK PLANE SETTING
   • Set work plane parameters, including coordinate, total size and grid size.
   REJECT LAST
   • **DONE** completes the creation of the line and starts the generation of a new line.
   • **EXIT** will terminate this function.

6.1.11 IDENTIFY

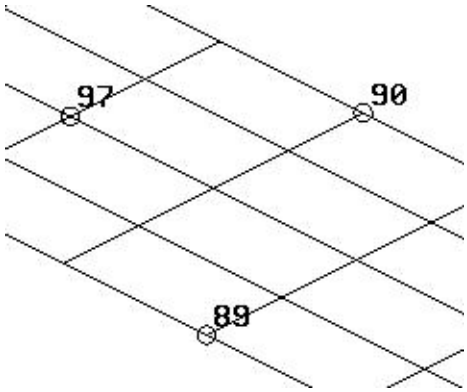
Description: This function allows the user to identify any node or point and its corresponding global location in X, Y, Z coordinates.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS
   ? ALL NODES
   KEY IN NODE NUMBER

NODE
POINT
EXIT

- If **ALL NODES** is selected, the numbers of the displayed nodes will appear on screen at their node locations (the first clear button in the DISPLAY PARAMETER OPTIONS WINDOW will remove the numbers)
- If **NODE** is selected, the number of the node nearest the cursor location will appear on screen. The following message will echo on the command line:
  > NODE xxxx X= x.xxx Y= y.yyy Z= z.zzz
- If **POINT** is selected, the number of the point nearest the cursor location will appear on screen. The following message will echo on the command line:
  > POINT xxxx X= x.xxx Y= y.yyy Z= z.zzz
- **EXIT** will send the user to the LINE menu.



6.1.12 MIRROR

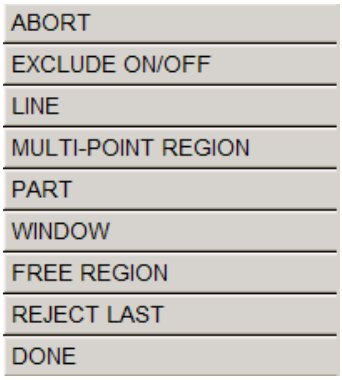
Description: This function allows the user to reflect a copy of desired lines across a local or global coordinate plane.

Usage:

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is now asking for a coordinate system to mirror the lines. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)

3. If the user selects YES, then VPG prompts:
   > PICK LINES TO MIRROR
   > SELECT LINES



4. Once desired lines have been selected, DONE takes the user to the next step.

5. VPG prompts:
   > ENTER PLANE OF MIRROR (XY, YZ, XZ)
   • The user may now select a local plane of symmetry in which to mirror lines.

6. VPG prompts:
   > INCLUDE MIRRORED LINES IN ORIGINAL PART? (Y/N)
   ? YES / NO
   • **YES** will place all selected lines into the ORIGINAL PART.
   • **NO** will place all selected lines into the current part.

6.1.13 OFFSET

Description: This function allows the user to copy and offset a selected line. The new line is generated in the UV plane according to a user-specified distance and direction.

Usage:

1. VPG prompts:
   > SELECT LINE TO OFFSET

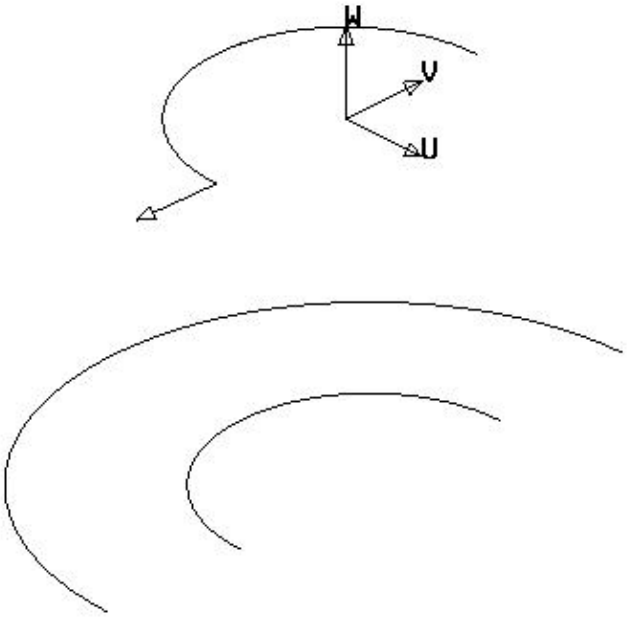
2. Once the desired lines have been selected, VPG prompts:
   > DEFINE PLANE FOR LINE OFFSET
   > PICK NODE/PT FOR ORIGIN
   • The user enters a coordinate system to define the plane in which the line lies. If the line does not lie in the global XY, XZ, or YZ plane, then the user selects three points on the line to define the plane. A thorough explanation of how to create a local coordinate system is covered in **Local Coordinate System**, Section 2.16.

3. Once the desired coordinate system is acquired, it will be displayed on the screen, and the prompt will read:
   > ACCEPT? (YES/NO)

4. If the user selects **YES**, then VPG prompts:
   > ENTER OFFSET DISTANCE IN UV PLANE OR -1 TO EXIT
   • The user enters a positive distance and an arrow appears at the beginning of the line.

5. VPG prompts:
   > ACCEPT OFFSET DIRECTION? (YES/NO)
   • **YES** creates a line in the displayed direction at the specified distance.

- **NO** creates a line opposite the displayed direction at the specified distance.



6.1.14 PROJECT

Description: This function projects lines and/or points onto selected surfaces.

Usage:

1. VPG prompts:
   > SELECT SURFACE

2. Once the desired surfaces are selected, click on **DONE**. VPG prompts the user to create/choose a coordinate system.
   > DEFINE PROJECTION VECTOR
   > PICK NODE/PT FOR ORIGIN

3. After accepting the coordinate system, VPG prompts:
   > SELECT ENTITY FOR PROJECTION:

4. Based on the selected entity, lines/points are projected onto the surface. If the point or points on the line do not intersect the surface along the global Z or W vector, VPG displays a message:
   > NO PROJECTION FOUND FOR XXXX POINT.

6.1.15 RESPACE LINE

Description: This function allows the user to re-space the number of points on the selected line equidistantly.

Usage:

1. VPG prompts:
   > SELECT LINE FOR RESPACING POINTS

2. Once the desired line is selected, VPG prompts:
   > ENTER NUMBER OF POINTS ON LINE OR E TO EXIT
   - Enter any integer greater than two. VPG includes the endpoints of the line as part of the total count of points on the line.

3. **EXIT** or **DONE** to exit this function.

6.1.16 REVERSE DIRECTION

Description: Each line has a direction defined from the starting point to the end point of the line. This function allows the user to reverse the line direction. Arrowheads on the line (presented during SHOW LINE, SPLIT, etc.) are identifiable points and also indicate the line direction.

Usage:

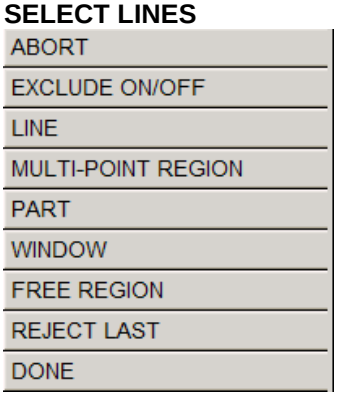
1. VPG prompts:
   > SELECT LINES
   - Once the line is selected, the direction is automatically reversed.

6.1.17 SCALE FACTOR

Description: This function allows the user to scale the selected lines or points with respect to any axis (global or local) using magnification factors in the X, Y, or Z (U, V, or W) directions. Entering a scale factor greater than 1 will magnify the line, whereas a scale factor less than 1 will reduce the length of the line.

Usage:

1. VPG prompts:
   > PICK LINES TO SCALE
   > SELECT LINES



   - Select **DONE** once the desired lines have been selected.

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG asks for a coordinate system to scale the lines and prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

3. Once the desired coordinate system is created, it will be displayed on the screen and the user will be prompted to accept or decline the system. If the user accepts, then VPG

prompts:

> **ENTER X, Y, Z SCALE FACTORS**

- The user selects the desired scale factors.

Example: The factors 1,-1,1 reflect the line size in the negative Y-direction (across the UV plane); the factors 0.5,0.5,0.5 reduce the lines by 0.5.

6.1.18 SECTION CUT

Description: This function will generate line segments along a user-defined UV plane at the intersection of selected lines intersecting the UV plane. The line section that is created will be included in the CURRENT PART (see PART CONTROL MENU).

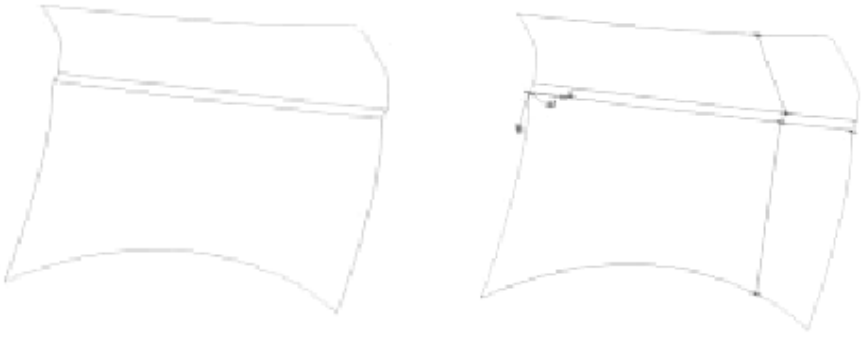
Usage:

1. VPG prompts:
   > **CREATE LOCAL COORDINATES**
   > **PICK NODE/PT FOR ORIGIN**
   - VPG is now asking for a coordinate system to create section lines. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

2. Once the desired coordinate system is acquired, it will be displayed on the screen.

3. If the user accepts the coordinate system, VPG prompts:
   > **DISTANCE FROM PLANE, W=?**
   - Users may now input a distance along the W-axis from origin of the selected coordinate system.
   - Any real value is a valid response.
   - The section line will be created at the distance W in the UV plane.

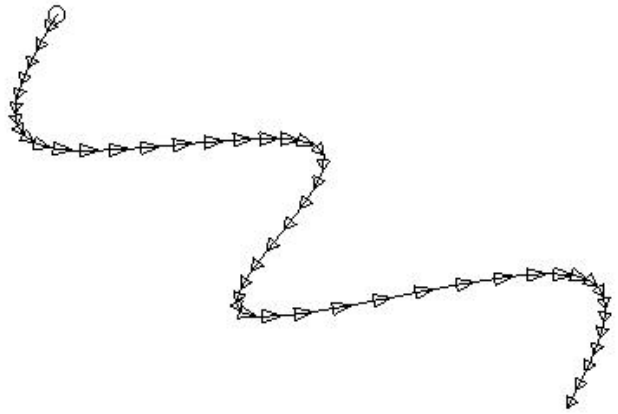
A section line is defined at a specific distance from the user defined local coordinate system.



4. VPG prompts:
   > **SELECT LINES**
   - **ABORT** exits the function without creating a section line.
     DONE will complete the section line and return the user to step 3.
     EXIT completes the section line and exits this function.

6.1.19 SHOW

Description: The user may identify any existing line or line direction using this function. The selected line is highlighted (the starting point is labeled with a circle) and subsequent points are represented with arrowheads oriented in the direction of the line.



The arrows indicate point locations on the line and line direction.

Usage:

1. VPG prompts :
   > SELECT LINES
   ? ABORT
   KEY IN LINE NO: enter the line No. to select a line
   LINE: select line with cursor
   DONE
   EXIT

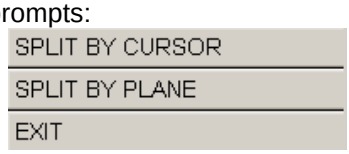
   • **EXIT** completes the show line and exits this function.

6.1.20 SPLIT (LINE)

Description: The user may split a line at a desired cursor location, at the nearest point, or at an intersection with another line.

Usage:

1. VPG prompts:



   • **SPLIT BY CURSOR** is selected, go to step 2, **SPLIT BY PLANE** is selected, go to step 4

2. VPG prompts:
   > PICK LINE TO SPLIT

3. VPG prompts:
   > PICK SPLIT LOCATION
   ? ABORT
   CURSOR LOCATION (anywhere the user clicks the mouse)
   LINE
   (The line will split at the location closest to the intersection of the

selected line.)
POINT
(The line will split at the selected point closest to the selected line.)
REJECT LAST

- **ABORT** will exit this function.

4. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

5. Once the desired coordinate system is acquired, it will be displayed on the screen.

6. If the user accepts the coordinate system, VPG prompts:
   > DISTANCE FROM PLANE, W=?
   - Users may now input a distance along the W-axis from origin of the selected coordinate system.
   - Any real value is a valid response.

7. VPG prompts:
   > PICK LINE TO SPLIT
   ? ABORT
   LINE The line will be split by UV offset plane
   REJECT LAST
   DONE
   EXIT

   - **ABORT** will exit this function.

6.1.21 TRANSFORM

Description: This function allows the user to translate or rotate the selected lines or points to a new location.

Usage:

1. VPG prompts:

   - If **LCS** is selected, go to step 2, if **POINT TO POINT** is selected, go to step 6

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

3. Once the coordinate system is determined, VPG prompts:
   ? MOVE

ROTATE
ABORT

- If **MOVE** is entered:
  > ENTER U, V, W INCREMENTS
  - i.e., 0,0,1000. This will move the selected line(s) 1000 units along the W-axis.
- If **ROTATE** is selected, the user enters an angle increment.

Note: All rotational commands are about the local W-axis or the global Z-axis.

4. VPG prompts:
   > SELECT ENTITY FOR TRANSFORM
   ? ABORT
   LINE
   POINT

5. After transformation, the following options are available:
   ? AGAIN
   - Repeats the previous transformation.
     LINE
     POINT
     REVERSE OPERATION
   - Reverses the previous transformations.
     DONE
     EXIT
   - EXIT will return to step 1.

6. VPG prompts:
   > SELECT ENTITY FOR TRANSFORM
   ? ABORT
   LINE
   POINT

7. After selecting lines or points, VPG prompts:
   > SELECT REFERENCE LOCATION
   ? ABORT
   KEY IN X, Y, Z
   NODE
   POINT
   REJECT LAST
   DONE

8. After selecting reference location, VPG prompts:
   > SELECT TARGET LOCATION
   ? ABORT
   CURSOR ON LINE
   INCREMENTAL XYZ
   KEY IN X, Y, Z
   NODE
   POINT
   REJECT LAST
   DONE

9. After transformation, the following options are available:
   ? AGAIN
   - Repeats the previous transformation.
     LINE

POINT
REVERSE OPERATION

- Reverses the previous transformations.

DONE
EXIT

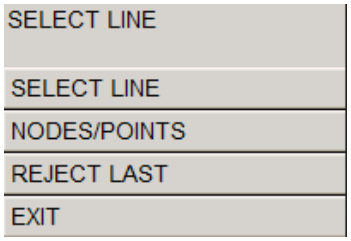
- EXIT will return to step 1.

6.1.22 BRIDGE LINE

Description: This function creates a spline curve to connect selected two lines.

Usage:

1. VPG prompts:
   > SELECT LINE BY PLACING THE CURSOR NEAR THE CONNECTING END
   > SELECT NODES/POINTS.BRIDGE LINE WILL BE CONNECTED TO THE LAST LOCATION



   ? SELECT LINE
   - Selects the first line.

   NODES/POINTS
   - Selects several nodes to create a line segment for the first line definition

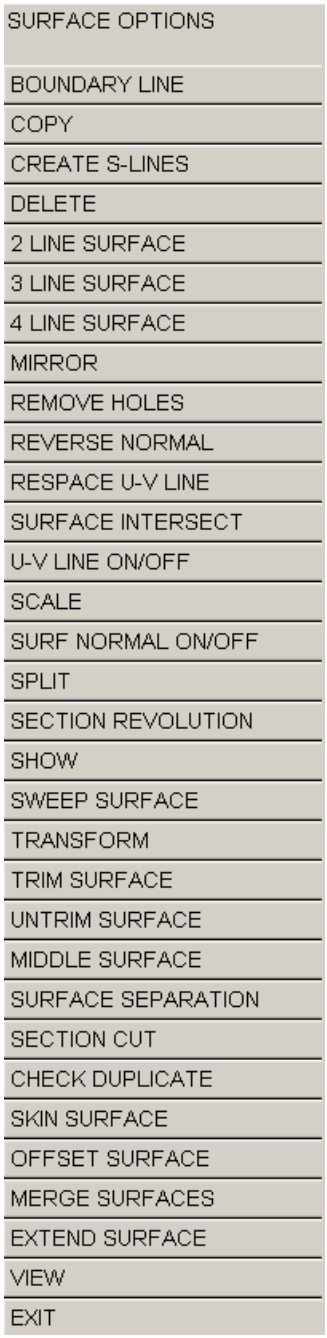
   REJECT LAST
   - Rejects last selected line

   EXIT

2. After selecting the first line, selects the second line, then a spline curve will be created to connect these two lines.
   - Lines created by method NODES/POINTS will be removed automatically after bridge line is created.

6.2 SURFACE

Commands in this menu are used to create and modify CAD surface data in VPG. VPG imports surface data and line data and converts it into VPG format. The CAD Import function in the FILE Menu is used to import externally created CAD data.



A detailed description of each function is given in the following sections.

The following are working definitions used throughout this section of VPG:

• **Domain:** The trimmed surface of the common region of the interior of the outer boundary and the exterior of the inner boundaries as well as the boundary curves.
• **Outer Boundary:** One boundary that is situated within the domain (the surface that is to be trimmed). In particular, it describes the boundary curve of the domain.
• **Inner Boundary:** Any number of inner boundaries including zero. The set of inner boundaries satisfies two criteria:

(a) The curves as well as their interiors are mutually disjointed.
(b) Each curve lies in the interior of the outer boundary.

6.2.1 BOUNDARY LINE

Description: This function creates a line on the boundary of a selected surface.

Usage:

1. VPG prompts the user to select a surface on the screen.
   > SELECT SURFACE FOR BOUNDARY LINE

2. VPG creates a boundary line around the selected surface and includes it in the current part.

Note: The boundary line should be split if it is intended for use with line meshing.

6.2.2 COPY

Description: This function copies selected surface(s) to a new location.

Usage:

1. VPG prompts the user to select surfaces to copy.

2. VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system. (Refer to Local Coordinate System, Section 2.16 for a thorough explanation of this procedure.)

3. Once the desired coordinate system has been defined, VPG prompts for the number of copies.

4. Once the number of copies has been entered, VPG prompts for the method of copy.
   > MOVE OR ROTATE (M/R)
   ? MOVE
   > ENTER U, V, W INCREMENTS
   • i.e., /0,0,1000/. This copies the surface(s) 1000 units along the W-axis or global Z.
   ROTATE
   • If ROTATE is selected, enter the angle increment (right-hand rule) about the W-axis. The remaining commands are the same as the MOVE option.
   ABORT

5. VPG prompts:
   > INCLUDE COPIED SURFACE IN ITS ORIGINAL PART. (Y/N)
   ? YES
   NO

6.2.3 CREATE S-LINES

Description: This function creates sectional line data on the selected surface.

Usage:

1. VPG prompts the user to select a surface on the screen.

2. Once the surface is selected, VPG places edge numbers on the surface and prompts the user to enter a number of section lines along edges 1 and 2.

3. VPG creates a set of section lines along edges 1 and 2. These lines are included in the current part.

6.2.4 DELETE

Description: This function deletes the selected surfaces from the database.

Usage:

1. VPG prompts the user to select a surface on the screen.
   > SELECT SUFACE
   - Surfaces will be highlighted as they are selected, Surfaces may be de-selected by selecting REJECT LAST from the menu, after selecting surfaces and click DONE, the selected surfaces are deleted.

6.2.5 2 LINE SURFACE

Description: This function generates surfaces between two selected lines. The orientation of the surface is independent of the direction of the lines. The created surface is included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > PICK LINE
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 2 LINE SURFACE function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 2 LINE SURFACE.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.
   - Once the lines have been selected and a surface is created

6.2.6 3 LINE SURFACE

Description: This function generates a surface in an area defined by 3 lines (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). The new surface is included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > SELECT 3 LINES

- LINE SEGMENT allows the user to combine a broken line while selecting lines for the 3 LINE SURFACE function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 3 LINE SURFACE.
- The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.
- Once the lines have been selected and a surface is created

Note: Select lines in a clockwise or counterclockwise direction.
Note: The direction of the surface normal dictates the direction of the plate element normals.

6.2.7 4 LINE SURFACE

Description: This function generates a surface in an area defined by 4 selected lines. The created surface is included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > SELECT LINES
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 4 LINE SURFACE function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 4 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.
   - Once the lines have been selected and a surface is created

Note: Select lines in a clockwise or counterclockwise direction.
Note: The direction of the surface normal dictates the direction of the plate element normals.

6.2.8 MIRROR

Description: Reflects a copy of desired surfaces across a local or global coordinate plane.

Usage:

1. VPG prompts the user to select a surface on the screen.

2. VPG prompts the user to select a coordinate system followed by the plane of the mirror. The user may create a local coordinate system using any method available in VPG.

3. The user then has the option of putting the surface in original part or current part.

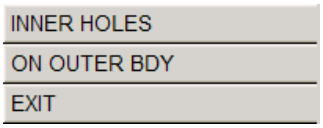
6.2.9 REMOVE HOLES

Description: This function allows the user to remove holes on the selected surface.

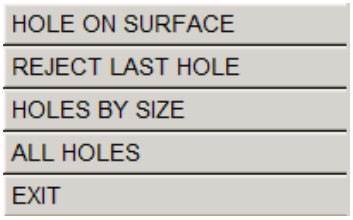
Usage:

1. VPG prompts the user to select surface for remove holes.

2. VPG prompts the user to select mode to remove holes on surface.



3. When selecting mode INNER HOLES, VPG prompts the user to select hole on surface to remove.



4. After selecting the hole, click EXIT, and the hole on surface is removed.

6.2.10 REVERSE NORMAL

Description: This function reverses the normal direction of the surface. The surface normal affects light source shading on some workstations. REVERSE NORMAL also controls the normal direction of the plate elements during meshing.

Usage:

1. VPG prompts:
   > SELECT SURFACE
   - The normal direction is changed after a surface is selected.

6.2.11 RESPACE UV LINE

Description: This function changes the UV line density of the selected surfaces.

Usage:

1. VPG prompts the user to select a surface on the screen.

2. VPG labels the edge numbers on the surface and prompts the user to enter the number of UV lines along edges 1 and 2.

3. VPG creates a set of UV lines along edges 1 and 2. These UV lines are included in the original part.

Note: This operation changes only the appearance of the selected surfaces. It does not change the integrity of the surface.

6.2.12 SURFACE INTERSECT

Description: This function creates a line at the intersection between two selected surfaces.

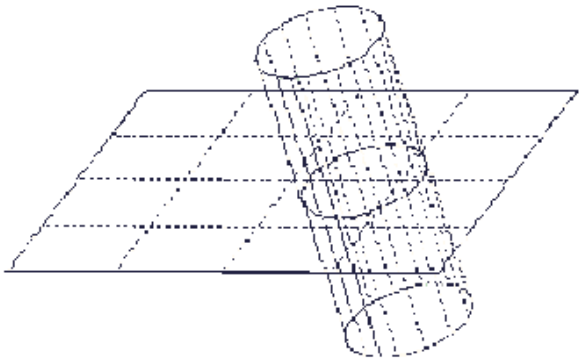
Usage:

1. VPG prompts the user to choose a surface.
   > SELECT THE FIRST SURFACE

2. After selecting the first surface, VPG prompts the user for the second surface.

> SELECT THE SECOND SURFACE

3. After selecting the second surface, VPG automatically creates a line at the intersection of the two surfaces. The newly created line is included in the current part.



6.2.13 UV LINE ON/OFF (toggle)

Description: This function controls the on and off display of UV lines on a surface.

Note: Toggling the UV lines off has no effect on the outcome of meshing, trimming, or splitting of surfaces.

6.2.14 SCALE

Description: This function allows the user to scale selected surfaces with respect to any axis (global or local) using magnification factors in the X, Y, or Z (U, V, or W) directions. A scale factor greater than 1 will magnify the surface, whereas a scale factor less than 1 will reduce the surface.

Usage:

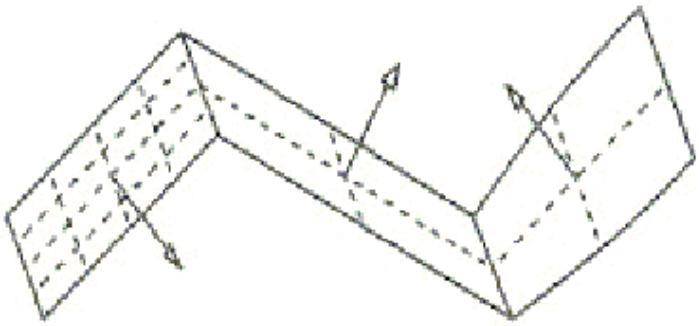
1. VPG prompts the user to select the surfaces.

2. Once the surfaces have been selected, VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system. (Refer to Local Coordinate System, Section 2.16, for a thorough explanation of this procedure.)

3. Once the desired coordinate system has been defined, VPG prompts:
   > ENTER X,Y,Z SCALE FACTOR, OR E TO EXIT
   - This scales the surface.

6.2.15 SURF NORMAL ON/OFF (toggle)

Description: This function controls the on and off display of normal vectors on a surface. The normal vector below is shown at the center of the surface.



6.2.16 SPLIT

Description: This function allows the user to split selected surfaces in several different ways. All the options for splitting surfaces are addressed below.

Usage:

1. VPG prompts to select surface.

2. Upon selection of the surface, VPG prompts:
   > SELECT OPTION TO SPLIT (DEFAULT IS SECTION LINE)
   • User selects a displayed section line to split a surface.
   ? TWO POINTS ON BOUNDARY
   > SELECT FIRST POINT ON BOUNDARY
   > SELECT SECOND POINT ON BOUNDARY
   • VPG splits the surface once the user selects the second boundary point.
   RESPACE U-V LINES
   > ENTER THE NUMBERS OF UV LINES: N1, N2
   • N1 and N2 cannot be greater than 199.
   • The user may choose a desired section line once the user selects the section lines for N1 and N2.
   SECTION U-V LINE
   > SELECT SECTION LINE TO SPLIT SURFACE
   SURFACE INTERSECTION
   > SELECT THE SECOND SURFACE
   • Once the user selects the second surface, the first surface splits at the intersection of the two surfaces. The user may split the second surface at the same intersection.
   SPLINE CURVE
   > SELECT NEXT POINT FOR SPLINE CURVE
   • The user may pick up to 500 locations on the surface in order to complete a spline curve.
   LINEAR SEGMENTS
   > SELECT NEXT POINT FOR LINEAR SEGMENT
   • The user may pick up to 500 locations on the surface to complete a piecewise linear curve.
   CURVE IN DATABASE
   > SELECT LINES
   • The user may select lines in database as split lines.
   OVERLAP SURFACE
   > SELECT AN OVERLAP SURFACE

- The user may select an overlap surface as split line.

DEFINE LCS

> PICK NODE/PT FOR ORIGIN

- This will lead user to create a coordinate system.

EXIT

6.2.17 SECTION REVOLUTION

Description: This function creates a surface by rotating a selected section line (the generatrix) up to 360º about a vector (the axis of revolution).

Usage:

1. VPG prompts:

> SELECT THE AXIS OF REVOLUTION

> SELECT THE FIRST POINT FOR THE AXIS

? ABORT

LINE (TWO POINTS)

- The user may select a line, defined by only two points, to be the axis of revolution.

TWO POINTS

- Default.

REJECT LAST POINT

UNDO

EXIT

- The user has been prompted to select the first of two points required to define the axis of revolution. Once the first point has been selected, VPG prompts:

> SELECT THE SECOND POINT FOR THE AXIS

- The user selects the second point, then VPG displays a vector representing the axis of revolution.

> SELECT A LINE AS THE GENERATRIX OF REVOLUTION

? LINE (default)

SEGMENT (allows user to combine line segments into one line)

REJECT (rejects last selection)

DONE

2. VPG prompts:

>ENTER START AND END ANGLE [DEFAULT: 0., 360.]

- The user may enter a desired start angle and end angle to generate a surface or press ENTER to generate a 360 degree angle.

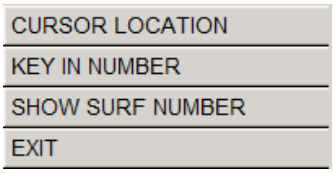
6.2.18 SHOW

Description: This function highlights the selected surface and UV lines in white color. Choosing CLEAR in the VPG DISPLAY PARAMETER OPTIONS MODULE removes highlights.

Usage:

1. VPG prompts:

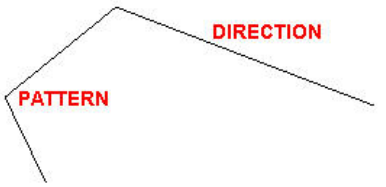
> SELECT SURFACE

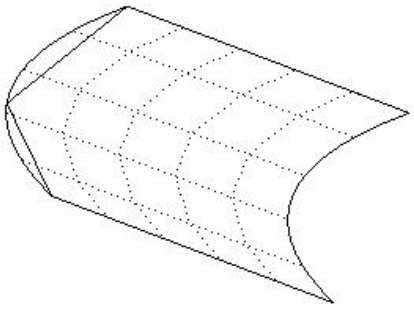


- **CURSOR LOCATION**, Select the surface in the graphics display window directly (default), the surface will be highlighted and ID will be shown.
- **KEY IN NUMBER**, Enter the surface number in the field; the surface will be highlighted and ID will be shown.
- **SHOW SURF NUMBER**, All surfaces ID of the displayed surfaces will be shown.
- **EXIT**

6.2.19 SWEEP SURFACE

Description: This function allows the user to create a surface via the selection of a section line and a direction line. The surface is generated along the length of the direction line.





Usage:

1. VPG prompts the user to select a sweep type:
   > SELECT SWEEP TYPE
   NORMAL SWEEP
   - The section lines remain perpendicular to the line of direction.

   RIGID SWEEP
   - The section lines remain parallel to the line of pattern.

2. VPG prompts the user to select a line.
   > SELECT A LINE FOR THE SWEEP PATTERN
   - The selected line will act as a constant cross section for the new surface.

3. Once the line has been selected, VPG prompts:
   > SELECT A LINE FOR THE SWEEP DIRECTION
   - The new surface will generate along the selected line and the user will return to step 2.

6.2.20 TRANSFORM

Description: This function allows the user to move or rotate a selected surface either locally or globally.

Usage:

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system.

2. Once the desired coordinate system has been defined, VPG prompts:
   > MOVE OR ROTATE (M/R)
   ? MOVE
   > ENTER U, V, W INCREMENTS
   - i.e. /0,0,1000/. This will transform the selected surface(s) 1000 units along the W-axis.

   ROTATE
   - If ROTATE is entered, enter angle increment (right hand rule) about the W-axis. The remaining commands are the same as the MOVE option.

   ABORT
   - User will now be prompted to select the surface to be transformed.

3. VPG prompts the user to select surfaces to transform. The user also has the option to select AGAIN or REVERSE to repeat the rotation or movement. Once the desired surfaces are selected, enter DONE to continue.

6.2.21 TRIM SURFACE

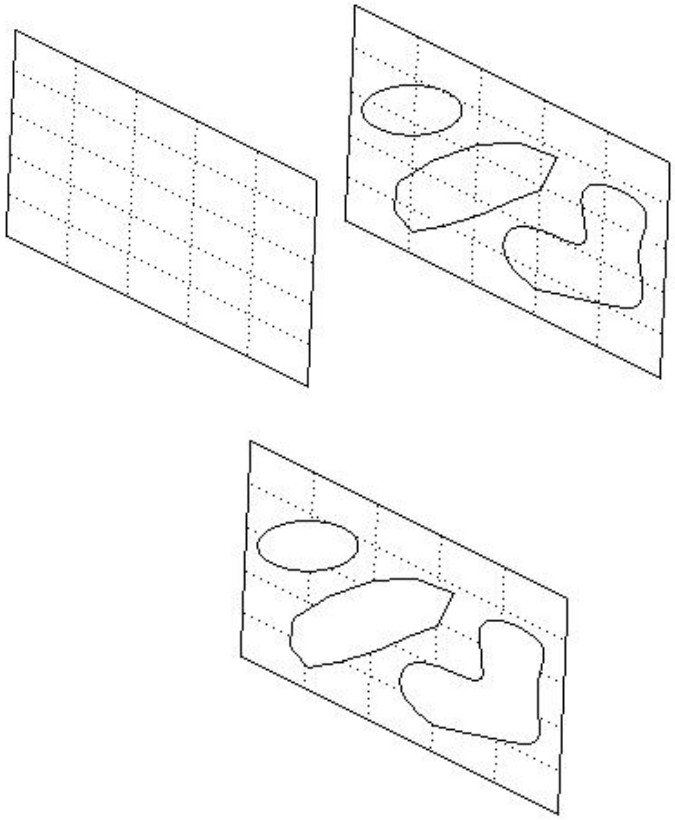
Description: This function defines curves for trimming edges and/or cutting holes on the selected surface. The defined curves may be poly-lines, b-spline lines, circles, piecewise linear segments, or intersecting lines of two surfaces.

Usage:

1. VPG prompts for a surface:
   > SELECT SURFACE FOR TRIM OPERATION

2. Upon selection of the surface, VPG prompts:
   > PLEASE SELECT OR DEFINE LINES
   ? SELECT LINE
   - Directly select a line to trim surface.

   CURSOR TO DEFINE LINE
   - After defining a line, user needs to select line as trim line.



-TRIM SURFACE: CIRCLE

Description: This function cuts circles on a selected surface. The user defines the location of the circle via mouse pick.

Usage:

1. VPG prompts:
   > SELECT THE CENTER OF THE CIRCLE
   - The user defines the center of the circle anywhere on the surface.

   > SELECT A POINT ON THE CIRCLE
   - The user may select a second point on the surface to define the radius of the circle. VPG proceeds to draw the circle.

2. The user should then select exit. VPG prompts:
   > SELECT A CLOSED CURVE AS OUTER BOUNDARY
   ? ABORT
   SURFACE BOUNDARY
   - Selects the outer, closed, surface boundary.

   CURVE ON SURFACE
   - The user may select an existing curve on the surface (open or closed) as the outer boundary of the trim operation.

   REJECT LAST CURVE
   - Deselects the curve selected for the trim operation.

   DONE
   EXIT

3. VPG prompts:
   > SELECT CLOSED CURVE AS INNER BOUNDARY
   ? ABORT
   CURVE ON SURFACE
   - The user may select an existing curve on the surface (open or closed) as the inner boundary of the trim circle operation.

   NO INNER BOUNDARY
   - VPG trims the surface of the outer boundary.

   REJECT LAST CURVE
   DONE
   EXIT

4. VPG then prompts:
   > DELETE THE ORIGINAL SURFACE. (Y/N)
   - **YES** generates a trimmed surface.
     NO generates a trimmed surface and retains a copy of the original surface.

5. The user is returned to step 1 of **TRIM SURFACE** command.

-TRIM SURFACE: INTERSECTION LINE

Description: This function allows the user to trim away the intersection of two surfaces.

Usage:

1. VPG prompts:
   > SELECT SURFACE FOR TRIM OPERATION
   Note: This surface is referred to as a "primary surface" in this section.
   > SELECT THE SECOND SURFACE
   - Once the user selects the second surface, VPG draws a boundary line between the two intersecting surfaces.
     Note: This surface is referred to as a "secondary surface" in this section.

2. Once the second surface has been selected and the user enters EXIT, VPG prompts:
   > SELECT A CLOSED CURVE AS OUTER BOUNDARY
   ? ABORT
   DONE
   EXIT
   SURFACE BOUNDARY
   - This creates the outer closed-surface boundary.

   CURVE ON SURFACE
   - This locates an existing curve on the surface (open or closed) as the outer boundary of the trim operation.

   REJECT LAST CURVE
   - This rejects the last curve selected for the trim operation.

3. VPG prompts:
   > SELECT CLOSED CURVE AS INNER BOUNDARY
   - This retains the surface between the outer boundary and the inner boundary during the trim operation. The inner boundary can be a line segment, open spline, closed spline, or circle.
   - **ABORT**
     DONE
     EXIT

NO INNER BOUNDARY
- This trims the outside surface of the outer boundary.

CURVE ON SURFACE
- The user may select an existing curve on the surface (open or closed) as the inner boundary of the trim operation.

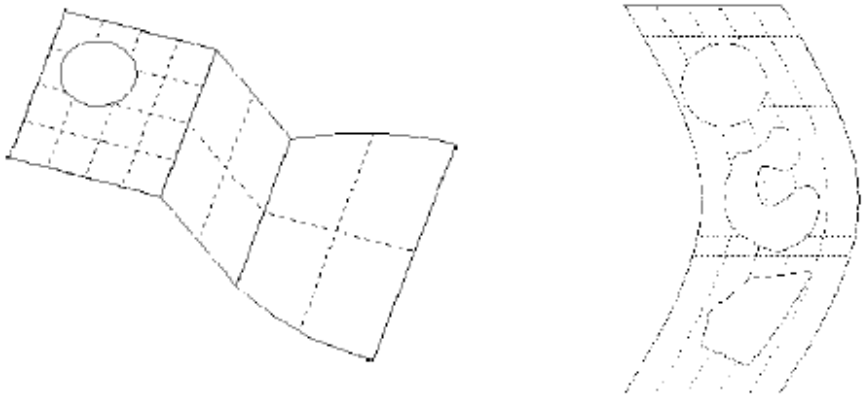
REJECT LAST CURVE
- This rejects the last curve selected for the trim operation.

4. VPG then prompts:
   > DELETE THE ORIGINAL SURFACE. (Y/N)
   - **YES** generates a trimmed surface.
     NO generates a trimmed surface and retains the original surface.

5. VPG returns the user to step 2 of the **TRIM SURFACE** command.

-TRIM SURFACE: OPEN CURVE

Description: This function defines an open curve on a selected surface. After determining the surface, type of curve, and points, this function connects the points leaving the curve open, with the first and last point selected as the starting and ending points of the curve.



-TRIM SURFACE: POINT

Description: POINT is the default option in the trim surface menu. Users may pick locations on the surface defining piecewise linear segments for trim operations.

-TRIM SURFACE: SPLINE CURVE

Description: This function cuts spline curves on a selected surface. The user defines the location of the spline via mouse pick.

Usage:

1. VPG prompts:
   > SELECT NEXT POINT FOR SPLINE CURVE.
   - Once the user has selected the points for the curve, then enters **DONE** (CLOSED), or enters OPEN CURVE to complete the spline.
     - **DONE** (CLOSED) defines a closed spline curve passing through all

the points selected.

- **OPEN CURVE** defines a spline curve passing through all the points selected, with the first and last point selected as the starting and ending points of the curve. The user then selects exit twice.

2. VPG prompts:

> SELECT A CLOSED CURVE AS OUTER BOUNDARY

- **ABORT**
  DONE
  EXIT
  SURFACE BOUNDARY
  - Selects the outer closed surface boundary.

  CURVE ON SURFACE
  - The user may select an existing curve on the surface (open or closed) as the outer boundary of the trim operation.

  REJECT LAST CURVE

3. VPG prompts:

> SELECT CLOSED CURVE AS INNER BOUNDARY

- The surface between the outer boundary and the inner boundary will be retained during the trim operation. The inner boundary can be a line segment, open spline, closed spline, or circle.

? ABORT
DONE
EXIT
NO INNER BOUNDARY
- The surface outside of the outer boundary will be trimmed.

CURVE ON SURFACE
- The user may select an existing curve on the surface (open or closed) as the inner boundary of the trim operation.

REJECT LAST CURVE
- Deselects the curve selected for the trim operation.

4. VPG then prompts:

> DELETE THE ORIGINAL SURFACE. (Y/N)

- **YES** generates a trimmed surface.
- **NO** generates a trimmed surface and retains the original surface.

5. The user is returned to step 2 of TRIM SURFACE command.

-TRIM SURFACE: REJECT LAST POINT

Description: This function deselects points when defining a trim line.

-TRIM SURFACE: DONE (CLOSED)

Description: This function defines a closed curve on a selected surface. After determining the surface, type of curve, and points, this function connects the points, closing the curve with the first and last points selected linked.

6.2.22 UNTRIM SURFACE

Description: This function removes all holes on the selected surface. Once the surface is selected, all holes on the surface are removed.

Usage:

1. VPG prompts to select surface.

2. Once the surface is selected, VPG prompts:
   > KEEP THE TRIM LINES?
   ? YES
   NO
   - If there is no hole on selected surface, VPG prompts:
     > SURFACE # IS UNTRIM SURFACE

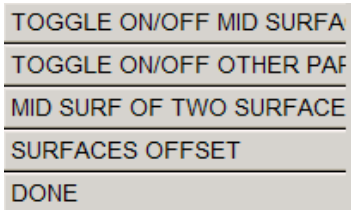
6.2.23 MIDDLE SURFACE

Description: This function creates a middle surface in a solid part and places this surface in a new part.

Usage:

1. VPG prompts:
   > SELECT SURFACE
   - . After surfaces are selected, VPG generates middle surfaces automatically into a new part and the thickness property of the middle surface will be added into this new part automatically.

2. VPG prompts:



   - **TOGGLE ON/OFF MID SURFACES**: display on/off generated middle surfaces.
   - **TOGGLE ON/OFF OTHER SURFACES**: display on/off surfaces except middle surfaces.
   - **MID SURF OF TWO SURFACES**: select two surfaces to generate middle surface.
   - **SURFACES OFFSET**: select surfaces to offset, offset distance is half of the part thick.
   - **DONE**: finish and exit this function.

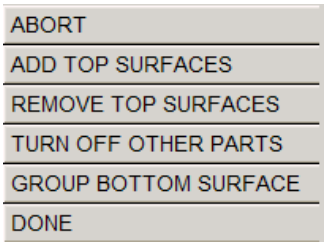
6.2.24 SURFACE SEPARATION

Description: This function separates the surfaces of a solid geometry into upper and lower surfaces, and places them in new parts.

Usage:

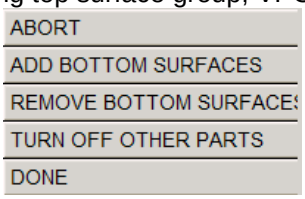
1. VPG prompts:
   > SELECT SURFACE

2. After selecting surfaces, VPG prompts:



- **ADD TOP SURFACES**: add bottom surfaces to top surfaces
- **REMOVE TOP SURFACES**: remove top surfaces to original surfaces.
- **TURN OFF OTHER PARTS**: don't display parts except part including top surfaces.
- **GROUP BOTTOM SURFACES**: turn to bottom surfaces operation.

3. After generating top surface group, VPG prompts:



- **ADD BOTTOM SURFACES**: add other surfaces to bottom surfaces.
- **REMOVE BOTTOM SURFACES**: remove bottom surfaces to original surfaces.
- **TURN OFF OTHER PARTS**: don't display parts except part including bottom surfaces.
- **DONE**: finish and exit this function.

6.2.25 SECTION CUT

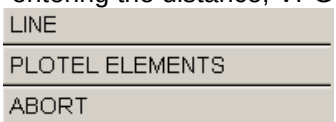
Description: This function creates a cross-section, which may be saved as a geometric line entity.

Usage:

1. VPG prompts:
   - **> CREATE LOCAL COORDINATES**
   - **> PICK NODE/PT FOR ORIGIN**
   - VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system.

2. Once the desired coordinate system has been defined, VPG prompts:
   - **> ENTER DISTANCE OF SECTION PLANE FROM W=0**

3. After entering the distance, VPG prompts:



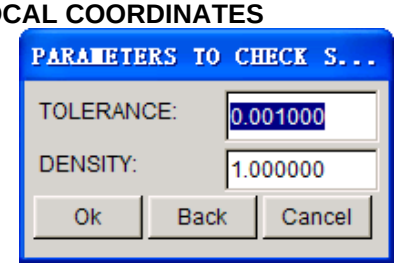
   - LINE: VPG prompts the user whether to add the line to a new part and enter the line split angle, and then the section line is created.
   - PLOTEL ELEMENTS: VPG prompts whether to add the plotel elements into a new part.

6.2.26 CHECK DUPLICATE

Description: This function checks duplicate surfaces in database.

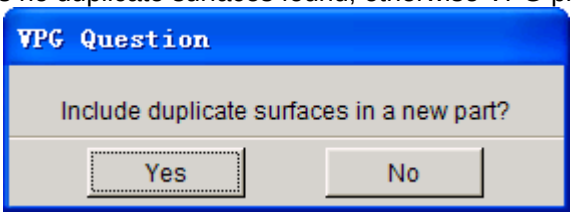
Usage:

1. VPG prompts:
   > CREATE LOCAL COORDINATES



- **TOLERANCE** search tolerance of the checked surfaces
- **DENSITY** sample point density level of the checked surfaces
- **Ok** accept current input
- **Back** return to last input
- **Cancel** exit this function without executing.

2. After defining above parameter, VPG begins to check duplicate surfaces in database, if there is no duplicate surfaces found, otherwise VPG prompts:



- **YES** duplicate surfaces are added to a new part; user needs to enter a name for this new part
  No duplicate surfaces are reserved in original parts

6.2.27 SKIN SURFACE

Description: This function creates a surface by several lines.

Usage:

1. VPG prompts:
   > SELECT LINES IN ORDER
   ? ABORT
   LINE
   - Select skin lines, this is the default option.

   REJECT
   - Reject last select lines

   DONE
   - generate surface after selecting lines, user can go on selecting lines to generate surfaces.

   EXIT

6.2.28 OFFSET SURFACE

Description: This function helps to offset surfaces along their normal or reverse normal directions for a distance.

Usage:

1. VPG prompts:
   > SELECT SURFACE

2. After selecting surfaces, VPG prompts:
   > IS THE OFFSET DIRECTION CORRECT?
   ? ABORT
   • Abort offset surface and exit this function.
   REVERSE
   • Reverse current offset direction.
   ACCEPT
   • Accept current highlighted offset direction.

3. After confirming the offset direction, VPG prompts:
   > ENTER OFFSET DISTANCE
   • enter distance value and click OK, the selected surfaces are offset.

6.2.29 MERGE SURFACES

Description: This function helps to merge two or more surfaces into one surface, user can reserve the original surfaces or not.
.

Usage:

1. VPG prompts:
   > SELECT SURFACE

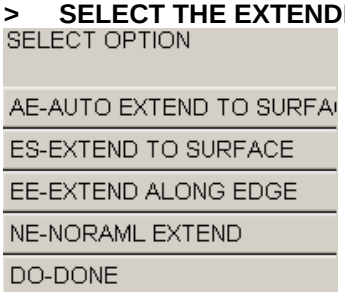
2. After selecting surfaces, VPG prompts:
   > DO YOU WANT TO DELETE ORIGINAL SURFACES?
   ? YES
   • The original surfaces will be deleted.
   NO
   • The original surfaces will be reserved.

6.2.30 EXTEND SURFACE

Description: This function helps to extend a surface; user can extend a surface to a target surface, or along an original surface edge, or perpendicular to an original surface edge.

Usage:

1. VPG prompts:
   > SELECT THE EXTENDING OPTION



AUTO EXTEND TO SURFACE

- Select a surface to extend; the program will automatically extend a surface till intersecting with another surface.

EXTEND TO SURFACE

- Select a surface to extend and a target surface, the program will automatically extend the first surface till the target surface.

EXTEND ALONG EDGE

- Select a surface to extend, and then select an edge of this surface, enter the extending distance; the surface will be extended along adjacent edges.

NORMAL EXTEND

- Select a surface to extend, and then select an edge of this surface, enter the extending distance; the surface will be extended perpendicular to the selected edge.

2. After selecting surfaces, VPG prompts:

> IS THE OFFSET DIRECTION CORRECT?

? ABORT

- Abort offset surface and exit this function.

REVERSE

- Reverse current offset direction.

ACCEPT

- Accept current highlighted offset direction.

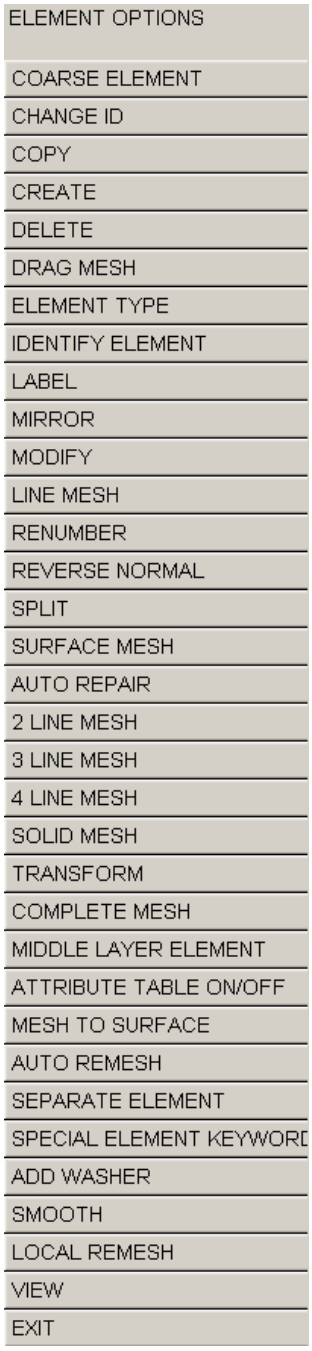
3. After confirming the offset direction, VPG prompts:

> ENTER OFFSET DISTANCE

- enter distance value and click OK, the selected surfaces are offset.

6.3 ELEMENT

The ELEMENT menu contains the commands to create and modify elements and to instantly mesh IGES surfaces. The 2, 3, and 4 LINE MESH commands create plate elements via line data selection, and the 6, 8, 9, and 12 LINE SOLID MESH commands create solid elements via line data selection. Elements can be modified to suit a particular model with the MODIFY, SPLIT, and COARSE ELEMENT commands. The commands in the ELEMENT menu are organized as follows:



VPG supports a wide range of element types for LS-DYNA and NASTRAN (refer to Appendix A: VPG Capabilities for F.E.A. Programs).

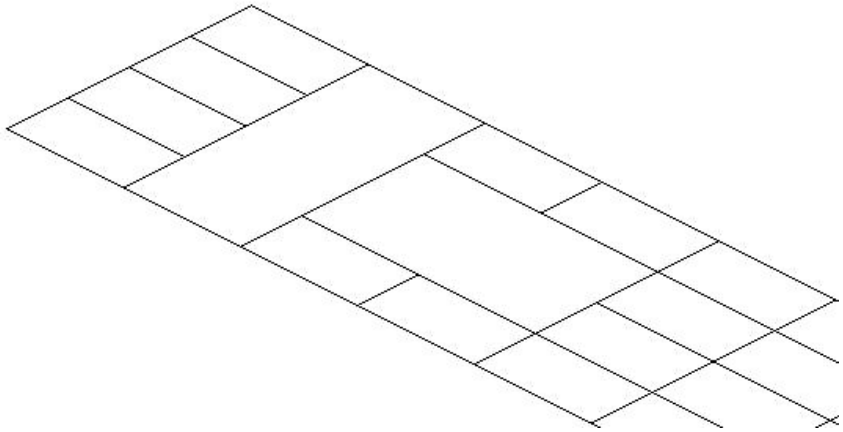
6.3.1 COARSE ELEMENT

Description: This function coarsens 2 CQUAD4 elements or 2 TRIA elements or a combination of a TRIA and a CQUAD4 into one element.

Usage:

1. VPG prompts the user to select 2, 3, or 4 elements:

2. Once desired elements have been selected:
   - DONE coarsens the selected elements and prompts for more.
   - ABORT or EXIT will exit this function.



The larger elements were created from groupings of four smaller elements.

6.3.2 CHANGE ID

Description: This function allows the user to change an existing element number. Parts do not need to be ON or CURRENT.

Usage:

1. VPG prompts the user to select elements:
   > SELECT ELEMENT FOR NEW ELEMENT NUMBER

2. Once the user selects a desired element, VPG prompts the user to enter the new element number:
   > ENTER NEW EL. NO. FOR xxxx OR [CR] TO IGNORE
   - xxxx is the original element number.

3. If the user enters a number that already exists, the prompt will read:
   > ELEMENT NUMBER yyyy ALREADY EXISTS, REQUEST DENIED
   > ENTER NEW EL. NO. FOR xxxx OR [CR] TO IGNORE

6.3.3 COPY

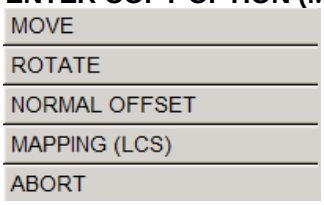
Description: This function allows the user to generate a set of duplicate elements. The user

may simultaneously translate, rotate, or offset (in the normal direction) the copied elements.

Usage:

1. VPG prompts the user to select the elements to be copied:
   > SELECT ELEMENTS

2. Once the desired elements have been selected, DONE continues to the next step, where VPG prompts:
   > ENTER COPY OPTION (M/R/N/P/A)



3. Then VPG prompts:
   > ENTER NUMBER OF COPIES

4. VPG prompts:
   > CREATE LOCAL COORDINATES
   PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system.

5. Once the coordinate system is determined, VPG prompts the user to select magnitudes for the transformation (respective to the selections made in step 4).
   - If MOVE was selected:
     > ENTER U, V, W, INCREMENTS
     - i.e., 0,0,1000. This will copy the selected element(s) 1000 units along the Z-axis.

   - If ROTATE was selected:
     > ENTER ANGLE INCREMENT
     - Enter the angle increment (right hand rule) about the W-axis.
   - If NORMAL offset was selected:
     > ENTER THICKNESS IN NORMAL DIRECTION
     - Thickness is the distance away from the plane of the elements in the direction of the element's normal.

6. VPG prompts:
   > INCLUDE COPIED ELEMENTS IN ITS ORIGINAL PART. (Y/N)
   ? YES
   NO
   - YES places all copied elements in their original part.
   - NO places all copied elements into the CURRENT PART.

-COPY: MAPPING (LCS)

Description: Mapping allows the user to copy and move or simply move selected elements from one user-defined coordinate system to another user-defined coordinate system along the UV plane.

Usage:

1. VPG prompts:
   > KEEP SELECTED ELEMENTS.
   - YES copies the selected elements and moves the copied elements to the new location.
   - NO removes the selected elements to the new location.

2. VPG prompts:
   > DEFINE FIRST LOCAL COORDINATE
   - This defines the orientation of the selected elements.

3. VPG prompts:
   > DEFINE SECOND LOCAL COORDINATE
   - This defines the new orientation for the selected elements.
   - VPG aligns the copied or moved elements according to their UV orientation.

6.3.4 CREATE

Description: This function creates elements of specific type by connecting selected nodes or points. The element type refers to the analysis program defined at the creation of the database. Depending on the analysis program, NASTRAN or LS-DYNA, VPG offers a list of element types and prompts the user to select one. Because the NASTRAN element types are more comprehensive, they are documented first in the following pages with the LS-DYNA elements following.

ELEMENT EQUIVALENCY TABLE		
NASTRAN/ABAQUS	SLS-DYNA / nonlinear codes	CMOLD/MOLDFLOW
BEAM ELEMENT (CBAR)	BEAM	PART ELEMENT
RIGID LINK (RBE2)	NODAL RIGID BODY	CHANNEL
PLOTEL ELEMENT	THICK SHELL	HOT RUNNER
PLATE ELEMENT	PLATE	COLD RUNNER
SOLID ELEMENT	SOLID	CONNECTOR
SPRING (CELAS2)	SPRING/DAMPER	
SPRING (CELAS1)	MASS	
RIGID BAR (RBAR)	SEATBELT	
MASS (CONM2)	SPOTWELD	
BEAM	JOINT	
TRUSS ELEMENT	INERTIA	
HOLLOW TUBE		
ISOPARAMETRIC SHELL		
RIGID BODY (RBE3)		
SPOTWELD		

-CREATE: BEAM ELEMENT (CBAR) (NASTRAN)

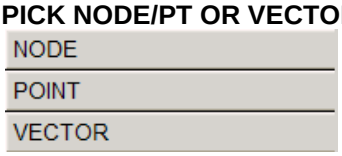
Description: This section covers the options for the BEAM ELEMENT subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER CBAR OFFSET VALUE

2. Once offset value has been entered, the prompt will read:
   > **PICK NODES/POINTS FOR CBAR ELEMENT**
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.

3. Once the desired nodes/points have been selected, VPG prompts:
   > **PICK NODE/PT OR VECTOR FOR CBAR ORIENTATION**



   - Up to three values may be entered to define a vector in the bar's local XY plane. The local Y-axis of the bar will be plotted at the first node of the bar element as it is generated.

4. If user enters **VECTOR**, then VPG prompts:
   > **ENTER CBAR VECTOR**
   - i.e., 0, 0, 1

-CREATE: RIGID LINK (RBE2) (NASTRAN)

Description: This section covers the options for the RIGID LINK subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > **ENTER DEPENDENT D.O.F. (DEFAULT 123456 OR –1 TO EXIT)**
   - Any combination of integers 1, 2, 3, 4, 5 and 6 may be entered as degrees of freedom.
   - Blanks or commas are not allowed between the degrees of freedom. i.e., 123456 = all D.O.F.

2. Next, VPG prompts for nodes/points for element:
   > **PICK NODES/POINTS**
   > **AT INDEPENDENT NODE**
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.

3. Once the desired node or point has been selected, VPG prompts:
   > **AT DEPENDENT NODE**
   - The user may select up to 200 dependent nodes.

4. Once the user picks the dependent node(s), DONE registers the selection and restarts the prompt cycle.

-CREATE: PLOTEL ELEMENT (NASTRAN)

Description: This section covers the options for the PLOTEL ELEMENT subsection of CREATE ELEMENTS. PLOTEL elements are used as the cross-section pattern for the DRAG MESH operation.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - An element will be created after two nodes/points are selected.
   - EXIT or ABORT will exit this function.

Note: PLOTEL elements may be output to a NASTRAN bulk data file. They are ignored for other analysis programs.

-CREATE: PLATE ELEMENT (NASTRAN)

Description: This section covers the options for the PLATE ELEMENT (quadrilateral and triangular) subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - A CQUAD4 ELEMENT requires 4 nodes or points to be selected.
   - A CTRIA3 requires 3 nodes or points to be selected, then an entry of DONE.
   - EXIT or ABORT will exit this function.

-CREATE: SOLID ELEMENT (NASTRAN)

Description: This section covers the options for the SOLID ELEMENT (cube, wedge, tetrahedron) subsection of CREATE ELEMENTS.

Usage:

Note: A solid element cannot be created in a part that has plate properties. The user must either create a new part and assign it solid properties or make a part current that contains solid properties.

1. VPG prompts:
   > PICK NODES/POINTS FOR SOLID ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three:
     CTETRA: Pick 4 nodes/points and type DONE.
     CPENTA: Pick 6 nodes/points (pick the second set of 3 nodes/points in the same order as the first set of 3 nodes/points) and type DONE.
     CHEXA: Pick 8 nodes/points (pick the second set of 4 nodes/points in the same order as the first set of 4 nodes/points). The element will be created.
   - EXIT or ABORT terminates this function.

-CREATE: SPRING ELEMENT (CELAS2) (NASTRAN)

Description: This section covers the options for the SPRING ELEMENT (NASTRAN CELAS2) sub-section of CREATE ELEMENTS.

Usage:

1. VPG prompts:

> ENTER SPRING DIRECTION (1 to 6) AT EACH END

- Two entries are required, one for each end of the spring (e.g., 1,1) for the "x" direction.
- Any number of spring elements may be created for the same connectivity depending upon the number of directions required.

  1 - translational along the global X-axis
  2 - translational along the global Y-axis
  3 - translational along the global Z-axis
  4 - rotational along the global X-axis
  5 - rotational along the global Y-axis
  6 - rotational along the global Z-axis

2. Once the desired direction has been entered for the spring, VPG prompts:
   > ENTER SPRING RATE
   - Any positive real number may be entered for the spring rate.

3. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - Select 2 nodes/points. The element will be created.
   - EXIT or ABORT terminates this function.

-CREATE: SPRING ELEMENT (CELAS 1) (NASTRAN)

Description: This section covers the options for the general SPRING ELEMENT subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER SPRING DIRECTION (1 to 6) AT EACH END
   - Two entries are required, one for each end of the spring (e.g., 1,1) for the "x" direction.
   - Any number of spring elements may be created for the same connectivity depending upon the number of directions required.

     1 - translational along the global X-axis
     2 - translational along the global Y-axis
     3 - translational along the global Z-axis
     4 - rotational along the global X-axis
     5 - rotational along the global Y-axis
     6 - rotational along the global Z-axis
   - Once the desired direction has been entered for the spring, VPG prompts the user to enter a property. If no properties exist in the database at the current time VPG prompts to create one.

2. VPG prompts:
   > PICK NODES/POINTS FOR CELAS1 ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - Select 2 nodes/points. The element will be created.
   - EXIT or ABORT will exit this function.

-CREATE: RIGID BAR (RBAR) (NASTRAN)

Description: This section covers the options for the RIGID BAR section of CREATE

ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER D.O.F. (CNA, CNB, CMA, CMB)
   - Any combination of integers 1, 2, 3, 4, 5 and 6 may be entered as degrees of freedom.
     1 - translational along the global X-axis
     2 - translational along the global Y-axis
     3 - translational along the global Z-axis
     4 - rotational along the global X-axis
     5 - rotational along the global Y-axis
     6 - rotational along the global Z-axis
   - Blanks or commas are not allowed between the degrees of freedom, i.e. 123,456,456,345

2. Once the desired D.O.F. has been entered the prompt will read:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - Once the desired nodes (2) or points (2) have been selected, the element will be created.
   - EXIT or ABORT will exit this function.

-CREATE: MASS (CONM2) (NASTRAN)

Description: This subsection covers the options for the MASS subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > DEFINE CONM2 ELEMENT COORDINATE SYSTEM

2. VPG prompts:
   > ENTER MASS VALUE AND MOMENTS OF INERTIA
   - VPG displays a pop up window with the following items: I11, I21, I22, I31, I32, I33 etc. = MOMENT OF INERTIA
   - The user may enter only the mass if desired. The moment of inertia is an optional entry.
   - The user should remember that the units of the CONM2's should be consistent with the units of the database, e.g., DATABASE (MM.) = CONM2 (KG.)

3. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - The elements will be created simultaneously upon selection of the desired nodes/points.
   - EXIT or ABORT will exit this function.

-CREATE: BEAM (CBEAM) (NASTRAN)

Description: This subsection covers the options for the BEAM (NASTRAN CBEAM) section of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER CBEAM OFFSET VALUE
   • VPG displays a pop up window of CBEAM OFFSET VALUE

2. Once the beam offset value has been entered the prompt will read:
   > PICK NODES/POINTS FOR CBEAM ELEMENT
   • To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.

3. Once the desired nodes/points have been selected, VPG prompts:
   > PICK NODE/PT OR VECTOR FOR CBEAM ORIENTATION

   • Up to three values may be entered to define a vector normal to the beam's local XY plane.

4. When user enters **VECTOR**, VPG prompts:
   > ENTER CBEAM VECTOR
   • i.e., 0,0,1

-CREATE: TRUSS ELEMENT (NASTRAN)

Description: This section covers the options for the TRUSS ELEMENT subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER NEW PROPERTY NAME OR "EX" TO EXIT
   • VPG prompts the user to enter a property name if no properties exist in the database at the current time. Once a name is entered, a pop-up window will appear with a list of properties to edit.

2. Once the property ID has been entered the prompt will read:
   > PICK NODES/POINTS FOR ELEMENT
   • To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   • An element will be created after two nodes/points are selected.
   • EXIT or ABORT will exit this function.

-CREATE: HOLLOW TUBE (NASTRAN)

Description: This section covers the options for the HOLLOW TUBE subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER NEW PROPERTY NAME OR "EX" TO EXIT
   • VPG prompts the user to enter a property name if no properties exist in the database at the current time. Once a name is entered, a pop-up window will

appear with a list of properties to edit.

2. Once the property ID has been entered the prompt will read:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three.
   - An element will be created after two nodes/points are selected.
   - EXIT or ABORT will exit this function.

-CREATE: ISOPARAMETRIC SHELL (NASTRAN)

Description: This section covers the options for the ISOPARAMETRIC SHELL subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - Generating isoparametric tri or quad elements require the user to select three nodes/points per side (6).

-CREATE: RIGID BODY (RBE3) (NASTRAN)

Description: This section covers the options for the RIGID BODY (RBE3) subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER COMPONENT NUMBERS (DEFAULT 123456 OR –1 TO EXIT)
   - Defines the degrees of freedom.

2. VPG prompts:
   > SELECT REFERENCE NODE FOR RBE3

3. VPG prompts:
   > ENTER WEIGHTING FACTOR WT 1 (REAL) OR "0" TO END
   > ENTER COMPONENT WITH WEIGHTING FACTOR (DEFAULT 123)
   > FOR NODE WITH WEIGHTING FACTOR .100E+01
   > SELECT INDEPENDENT NODES

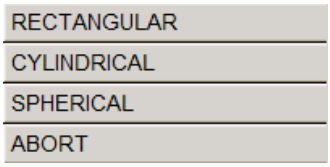
4. Step 3 is repeated until the user selects DONE or ABORT.

-CREATE: SPOTWELD (NASTRAN)

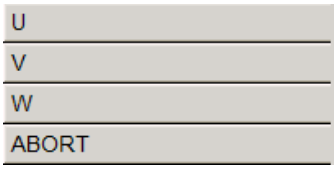
Description: This function allows the user to create local coordinate systems to be attached to the RBAR or RBE2 elements for the application of weld study. Two options are provided. The first option creates the RBAR and attaches the local coordinate systems at each node automatically. The second option attaches the local coordinate system automatically to the existing RBAR elements.

Usage:

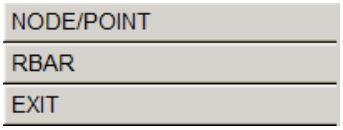
1. VPG prompts:
   > SELECT COORDINATE SYSTEM TYPE



2. VPG prompts:
   > SELECT LOCAL AXIS ALONG RBAR/RBE2



3. VPG prompts:
   > SELECT OPTION TO CREATE SPOTWELD



4. VPG prompts:
   > SELECT NODE/POINT FOR INDEPENDENT NODE
   - This step is repeated once, and the SPOTWELD is defined.
   - Once the SPOTWELD is defined, its coordinate system is displayed.

-CREATE: BEAM (LS-DYNA)

Description: This function defines a beam element. This section covers the options for the BEAM subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT

2. Select two nodes (points), VPG prompts:
   > SELECT NODES/POINTS FOR BEAM ORIENTATION
   - Select a node (point), and a beam element is created.

-CREATE: NODAL RIGID BODY (LS-DYNA)

Description: This function defines a NODAL RIGID BODY element between multiple nodes. The first selected node serves as a reference point; all additional nodes are connected to this first node (usually the center of the rigid body) for display purposes.

Usage:

1. VPG prompts:
   > SELECT THE FIRST NODE FOR RIGID BODY
   - The first node is selected.
   - Nodes can be selected by cursor pick, drag window, multi-point region, or part.

   > SELECT NODES
   - The second and subsequent nodes are selected until the user selects DONE.
   - The rigid body is displayed with double lines with an "R" alongside it.

-CREATE: PLATE (LS-DYNA)

Description: This function defines a triangle or quadrangle plate element. This section covers the options for the PLATE subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - Select three nodes, if click DONE, a triangle plate element is created, if go on selecting the fourth node, a quadrangle element is created.

-CREATE: SOLID (LS-DYNA)

Description: This function defines a tetrahedron or hexahedron element. This section covers the options for the SOLID subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - Select four nodes, if click DONE, a tetrahedron element is created, if go on selecting nodes, a hexahedron element is created after the eighth node is selected.

-CREATE: THICK SHELL (LS-DYNA)

Description: This section covers the options for the THICK SHELL subsection of CREATE ELEMENTS.

Note: A solid element cannot be created in a part that has plate properties. The user must either create a new part and assign it solid properties or make a part current that contains solid properties.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements, the user may select a node, point, keyboard entry, or any combination of the three:
   - EXIT or ABORT terminates this function.

-CREATE: SPRING/DAMPER (LS-DYNA)

Description: This function defines a spring/damper element between two nodes/points. It is recommended that the user select the ATTRIBUTE TABLE ON/OFF in the element options menu prior to creating a spring/damper. This will pop-up the ELEMENT DISCRETE CARD for user input.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - This step is repeated once.
   - When the second node/point is defined, VPG displays the SPRING/DAMPER element.

-CREATE: MASS (LS-DYNA)

Description: This function defines a lumped mass element on a node. This section covers the options for the MASS subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER LUMPED MASS VALUE

2. After entering a value, VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - Select a node, mass element will be created.

-CREATE: SEATBELT (2 NODES) (LS-DYNA)

Description: This function defines a seatbelt element between two nodes/points. Other seatbelt functions such as ACCELEROMETER, RETRACTOR, SENSOR, etc. are located in the DYNA MISCELLANEOUS menu.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - This step is repeated once.
   - When the second node/point is defined, VPG displays the SEATBELT element.

2. Once the seatbelt elements have been created it is recommended that the user enter the DYNA MISC. menu in the PRE PROCESSOR menu and select SEATBELT to define the accelerometer, pretensioner, retractor, sensor, and slipring.

-CREATE: SEATBELT (4 NODES) (LS-DYNA)

Description: This function defines a shell type seatbelt element with 4 nodes/points. This section covers the options for the SEATBELT (4 NODES) subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - When the fourth node/point is defined, VPG displays the SEATBELT element.

-CREATE: JOINT (LS-DYNA)

Description: This function allows the user to create joints for DYNA analysis. All types of joints are supported. The procedure to create joints has been simplified to create the coincident nodes for the second rigid body. The program first prompts for the Relative Penalty Stiffness (default=1.0). The program next prompts the user to select the first part (rigid body with material type 20) for the joint. If no material is defined for the selected part, the program prompts a warning message: "NO MATERIAL ASSIGNED TO SELECTED PART". The program then prompts the user to select necessary nodes in the first rigid body to define the orientation of the joint. The requirements for each type of joint are listed in the following table:

JOINT TYPE	NUMBER OF NODES	ORIENTATION	IDENTIFIER SYMBOL
Spherical	1 (N1)	Center	SJ

Revolute	2 (N1,N3)	Axis of Rotation	RJ
Cylindrical	2 (N1,N3)	Axis of Cylinder	CJ
Planar	2 (N1,N2)	Normal of Plane	PJ
Universal	3 (N1,N3,N4)	Center, End Points of the Perpendicular Axes	UJ
Translational	3 (N1,N3,N5)	Direction & Plane of Translation	TJ
Locking	3 (N1,N3,N5)	Axis & Plane	LJ

The program then prompts the user to select the second part (rigid body with material type 20) for the joint. All the required coincident nodes on the second part will be created and added to the second rigid body as CONSTRAINED_EXTRA_NODES_NODES. The following will take create locking as example:

Usage:

1. VPG prompts:
   >SELECT JOINT TYPE

2. VPG prompts:
   >ENTER RELATIVE PENALTY STIFFNESS (1.0)

3. VPG prompts:
   >SELECT 1ST PART (RIGID BODY ONLY) FOR JOINT
   >PICK AN ELEMENT OR PART NAME OF A PART

4. After part is selected, VPG prompts:
   >PICK NODE ON RIGID BODY FOR THE JOINT

5. VPG prompts:
   > PICK 2ND NODE ON RIGID BODY TO DEFINE THE 1ST AXIS OF JOINT

6. VPG prompts:
   >PICK 3RD NODE ON RIGID BODY TO DEFINE THE PLANE OF THE JOINT

-CREATE: PLOTEL ELEMENT (LS-DYNA)

Description: This section covers the options for the PLOTEL ELEMENT subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - This step is repeated until user enters ABORT or EXIT.

-CREATE: INERTIA (LS-DYNA)

Description: This section covers the options for the INERTIA subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts to define local coordinates

2. After local coordinates are defined, VPG prompts:
   > ENTER MASS INERTIA: Ixx, Ixy, Ixz, Iyy, Iyz & Izz

3. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT

-CREATE: SPH (LS-DYNA)

Description: This function defines a SPH element on a node. This section covers the options for the SPH subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER LUMPED MASS VALUE

2. After entering a value, VPG prompts:
   > PICK NODES
   - Select nodes and click DONE, SPH elements will be created.

-CREATE: CONST. NODE SET (LS-DYNA)

Description: This section covers the options for the CONST. NODE SET subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER APPLICABLE DOF: 1 TO 7, TYPE 0 TO EXIT

2. VPG prompts:
   > SELECT THE FIRST NODE FOR CONST. NODE SET

3. VPG prompts:
   > SELECT NODES

-CREATE: CONST. INTERPOLATION (LS-DYNA)

Description: This section covers the options for the CONST. INTERPOLATION subsection of CREATE ELEMENTS.

Usage:

1. VPG prompts:
   > ENTER COMPONENT NUMBERS (DEFAULT 123456 OR –1 TO EXIT)
   - Defines the degrees of freedom.

2. VPG prompts:
   > SELECT REFERENCE NODE FOR CONST. INTERPOLATION

3. VPG prompts:
   > ENTER COMPONENT WITH WEIGHTING FACTOR (DEFAULT 123456)
   > ENTER WEIGHTING FACTOR WT 1 (REAL) OR "0" TO END
   > SELECT INDEPENDENT NODES FOR CONST. INTERPOLATION

4. Step 3 is repeated until the user selects DONE or ABORT.

-CREATE: ALDS (LS-DYNA)

Description: This section covers the options for the ALDS subsection of CREATE ELEMENTS. This function will create ALDS (Arbitrary Location Deformable Spotweld) elements.

Usage:,

1. After define the section beam card, VPG prompts:
   > SELECT MASTER PARTS
   > SELECT AN ELEMENT OR PART NAME OF A PART.

2. After selecting parts, click EXIT, VPG prompts:
   > PICK ALDS LOCATION
   • Click cursor on the screen, ALDS is created. The contact between ALDS and elements they connect are created automatically.

-CREATE: MAT100 BEAM SPOTWELD (LS-DYNA)

Description: This section covers the options for the MAT100 BEAM SPOTWELD subsection of CREATE ELEMENTS.

Usage:

1. After define the section beam and mat100 card, VPG prompts:
   > SELECT MASTER PARTS
   > SELECT AN ELEMENT OR PART NAME OF A PART.

2. After selecting parts, click EXIT, VPG prompts:
   > PICK MAT 100 SPOTWELD LOCATION
   • Click cursor on the screen, MAT100 beam spotweld is created. The contact between spotwelds and elements they connect are created automatically.

-CREATE: MAT100 BRICK SPOTWELD (LS-DYNA)

Description: This section covers the options for the MAT100 BRICK SPOTWELD subsection of CREATE ELEMENTS.

Usage:

1. After define the section solid and mat100 card, VPG prompts:
   > PLEASE ENTER THE DIAMETER OF SPOTWELD

2. After entering the value, VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART.

3. After selecting parts, click EXIT, VPG prompts:
   > PICK SPOTWELD LOCATION
   • Click cursor on the screen, MAT100 brick spotweld is created. The contact

between spotwelds and elements they connect are created automatically.

-CREATE: MAT100 BY VIP FILE (LS-DYNA)

Description: This section covers the options for the MAT100 BY VIP FILE subsection of CREATE ELEMENTS. The *.vip to be read includes information about spotweld location.

Usage:

1. VPG prompts to select spotweld type as beam or solid.

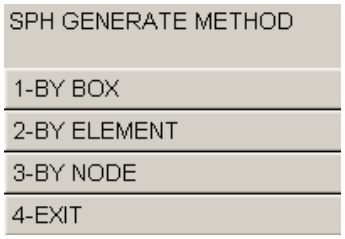
2. After selecting the spotweld type, a file dialogue pops up, open a *.vip file, MAT100 spotweld is created. The contact between spotwelds and elements they connect are created automatically.

-CREATE: SPH GENERATE METHOD (LS-DYNA)

Description: This section covers the options for the SPH GENERATE METHOD subsection of CREATE ELEMENTS.

Usage:

1. VPG offers three methods to create SPH elements:



2. If BY BOX is selected, VPG prompts:
   - **> CREATE LOCAL COORDINATES.**
   - After creating a coordinate, VPG prompts:
   - **> ENTER BOX EDGE LENGTH ALONG LOCAL X, Y, Z AXIS**
   - After entering the values, VPG prompts:
   - **> ENTER NO. OF SPH ALONG LOCAL X, Y, Z AXIS**
   - After entering the values, VPG prompts:
   - **> ENTER LUMPED MASS VALUE**
   - After entering the values, VPG prompts:
   - **> INCLUDE SPH ELEMENT INTO NEW PART (YES/NO)**
   - **? YES**
     - SPH elements will be included into a new created part

     NO
     - SPH elements will be included into current part
   - **> ACCEPT (YES/NO)**
   - **? YES**
     - SPH elements will be created

     NO
     - SPH elements will not be created.

3. If BY ELEMENT is selected, VPG prompts:
   - **> SELECT SHELL OR SOLID TO CREATE SPH AT THEIR CENTER**
   - **> SELECT ELEMENTS**
   - After selecting elements, VPG prompts:

> ENTER LUMPED MASS VALUE
- After entering the values, VPG prompts:

> INCLUDE SPH ELEMENT INTO NEW PART (YES/NO)
? YES
- SPH elements will be included into a new created part

NO
- SPH elements will be included into current part

> ACCEPT (YES/NO)
? YES
- SPH elements will be created

NO
- SPH elements will not be created.

4. If BY NODE is selected, VPG prompts:

> ENTER LUMPED MASS VALUE
- After entering the values, VPG prompts:

> SELECT NODES
- SPH elements are created at the location of the selected nodes.

-CREATE: SHELL ON SOLIDS (LS-DYNA)

Description: This section covers the options for the CREATE SHELL ON SOLIDS subsection of CREATE ELEMENTS.

Usage:

1. VPG checks free solid faces and prompts whether to create new part to include new created shell elements.
   - If click OK, new created shell elements will be included into this new created part.
   - If click CANCEL, a part list will pop up, new created shell elements will be included into selected part, if click EXIT directly, no shell elements will be created.

6.3.5 DELETE

Description: This function allows the user to delete elements using several different functions.

Usage:

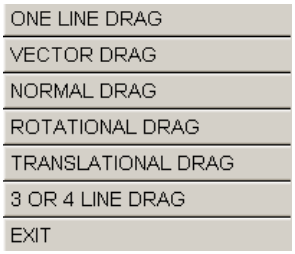
1. VPG prompts:

> SELECT ELEMENTS
- The elements selected and the nodes belonging to the elements will be deleted, if user wants to keep those unreferenced nodes, user needs to tick off the AUTOMATICALLY DELETE NODES WHEN DELETING ELEMENTS option on the SETUP DEFAULT deck.

6.3.6 DRAG MESH

Description: This function offers five methods to extrude elements from a cross-section of pattern elements. See the following figure for the five methods:

Usage:



1. VPG prompts:
   > SELECT DRAG MESH OPTION
   ? ONE LINE DRAG
   - Allows user to drag 1D elements into 2D elements (beam to plate) or 2D elements into 3D elements (plate to solid) along a selected line.

   VECTOR DRAG
   - Allows user to drag 1D elements into 2D elements (beam to plate) or 2D elements into 3D elements (plate to solid) along a selected vector.

   NORMAL DRAG
   - Extrudes plate elements into solid elements normal to the selected plate elements.

   ROTATIONAL DRAG
   - Extrudes beam elements (to plate elements) or plate elements (to solid elements) along the axis of rotation.

   TRANSLATIONAL DRAG
   - Extrudes beam elements (to plate elements) or plate elements (to solid elements) along W-axis.

   3 OR 4 LINE DRAG
   - Extrudes beam elements (to plate elements) or plate elements (to solid elements) along 3 or 4 selected lines.

   EXIT

   The following sections cover the details for each command.

-DRAG MESH: ONE LINE DRAG

Description: This section covers the options for the ONE LINE DRAG subsection of DRAG MESH. This function extrudes the selected elements along the selected line to form the higher order elements (i.e., 1D elements <PLOTEL, beams> are extruded to form plates, and plate elements are extruded to form solid elements).

Usage:

1. VPG prompts:

   ELEMENT: select elements to drag
   LINE: after a line is selected, VPG will prompt user to enter the number of elements on the selected line, temporary 1D elements will be created on this line.

2. Then VPG prompts:
   > SELECT CONTROL LINE FOR DRAG MESH
   - Number of points on the selected drag line dictates the number of sections

created.
- Line direction must point in the direction of the intended mesh.

3. Once the line is selected, the mesh is created. VPG prompts:
   > ACCEPT DRAG MESH? (Y/N)

-DRAG MESH: VECTOR DRAG

Description: This section covers the options for the VECTOR DRAG subsection of DRAG MESH. This function extrudes the selected elements along a vector to form the higher order elements (i.e., 1D elements <PLOTEL, beams> are extruded to form plates, and plate elements are extruded to form solid elements).

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. Then VPG prompts:
   > DEFINE TAIL OF VECTOR OR GLOBAL X/Y/Z AS VECTOR

3. Once the vector is defined. VPG prompts:
   > ENTER THE DRAG LENGTH ALONG THE VECTOR. DEFAULT: VECTOR LENGTH

4. After entering the length. VPG prompts:
   > ENTER NUMBER OF LAYERS THRU THICKNESS. (DEFAULT: 1)

5. Once the layer number is entered. VPG prompts:
   > ACCEPT DRAG MESH? (Y/N)

-DRAG MESH: NORMAL DRAG

Description: This section covers the options for the NORMAL DRAG subsection of DRAG MESH. The normal drag function extrudes the selected plate elements in the normal direction to create solid elements.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. Then VPG prompts:
   > ENTER THICKNESS OF NORMAL DRAG
   - This is the total thickness of the drag operation.

   > ENTER NUMBER OF LAYERS THRU THICKNESS (DEFAULT: 1)
   - This is the thickness of each layer of elements.

   > ACCEPT EXTRUDE MESH? (Y/N)

Note: Plate element normals should be consistent prior to the execution of NORMAL DRAG.

-DRAG MESH: ROTATIONAL DRAG

Description: This section covers the options for the ROTATIONAL DRAG subsection of DRAG MESH. This function extrudes the selected elements along the axis of rotation to form the higher order elements (i.e., 1D elements <PLOTEL, beams> are extruded to form plates, and plate elements are extruded to form solid elements).

Usage:

1. VPG prompts to select elements.

2. VPG prompts to define the axis of rotation.

3. VPG prompts:
   > ENTER ROTATION ANGLE ABOUT W-AXIS OR 0 TO EXIT

4. VPG prompts:
   > ENTER NUMBER OF LAYERS ABOUT W-AXIS OR 0 TO EXIT

5. Once the number of layers is entered, the mesh is created. VPG prompts:
   > ACCEPT MESH? (YES/NO/REMESH)

-DRAG MESH: TRANSLATIONAL DRAG

Description: This section covers the options for the TRANSLATIONAL DRAG subsection of DRAG MESH. This function extrudes the selected elements along the W-axis to form the higher order elements (i.e., 1D elements <PLOTEL, beams> are extruded to form plates, and plate elements are extruded to form solid elements).

Usage:

1. VPG prompts to select elements.

2. VPG prompts to create local coordinates.

3. Once local coordinates are defined, VPG prompts:
   > ENTER DRAG DISTANCE ALONG W-AXIS OR 0 TO EXIT

4. Then VPG prompts:
   > ENTER NUMBER OF LAYERS ABOUT W-AXIS OR 0 TO EXIT

5. Once the number of layers is entered, the mesh is created. VPG prompts:
   > ACCEPT MESH? (YES/NO/REMESH)

-DRAG MESH: 3 OR 4 LINE DRAG

Description: This section describes the options for the 3 OR 4 LINE DRAG subsection of DRAG MESH. The line drag commands extrude the selected plate elements to create solid elements bounded by 3 or 4 lines in space.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

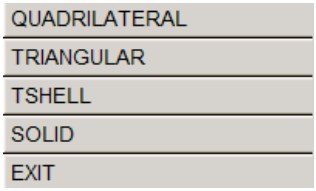
2. VPG prompts:
   > SELECT 3 (THEN PRESS D) OR 4 CONTROL LINES
   - The user must select the 4 volume lines in clockwise or counterclockwise order.

3. Once the desired lines are selected, VPG prompts:
   > ENTER NUMBER OF LAYERS OF SOLID ELEMENTS
   ? ACCEPT MESH. (Y/N)

- Line direction is not important during 4 LINE DRAG, but the lines should be of reasonable length to form the desired shape. The control lines do not have to pass through the nodes of the plate elements.

6.3.7 ELEMENT TYPE

Description: Defines element type prior to utilizing the 2, 3, 4 LINE MESH or SURFACE MESH. Element types are as following:



Usage: after finishing this function, elements created in 2, 3, 4 LINE MESH will be the element type selected.

6.3.8 IDENTIFY ELEMENT

Description: This function allows the user to identify an element and its nodes by cursor selection.

Usage:

1. VPG prompts:
   > SELECT ELEMENT BY CURSOR
   - VPG identifies the element's respective type, number, part, and nodes. The selected element will be highlighted, and element and node numbers will be displayed in the dialogue window.
   - The user also has the option to key in the number of a desired element to identify its location and characteristics.
   - It will also identify:

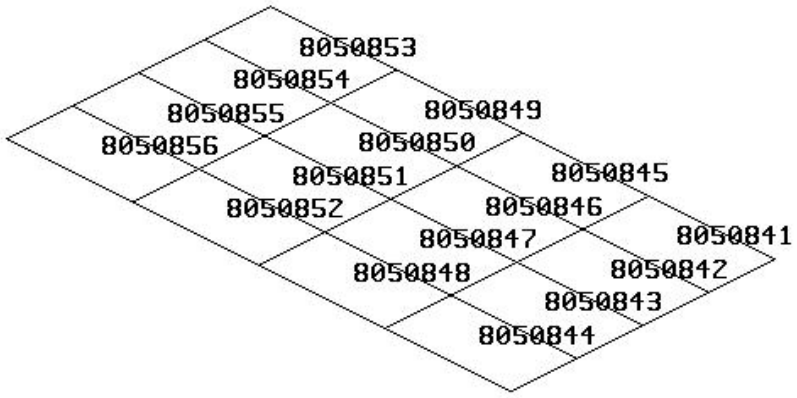
NASTRAN ELEMENT TYPE	DYNA ELEMENT TYPE	CMOLD/MOLDFLOW
CBAR	BEAM	PLATE
RBE2	RIGID EL	COLD RUNNER
PLOTEL	PLOTEL	HOT RUNNER
PLATE	PLATE	CONNECTOR
SOLID	SOLID	
CELAS2	SPRING	
CELAS1	SPOTWELD	
RBAR	MASS	
CTUBE	JOINT	
ISO-SHL	T-SHELL	
RBE3	SEATBELT	

6.3.9 LABEL

Description: This function toggles the element label on or off. When LABEL is executed and parts are turned ON, the screen displays the elements and the element ID numbers in the database.

Usage:

1. Depending on the current state of the toggle, VPG prompts either:
   > ELEMENT NUMBER WILL BE LABELED or
   > ELEMENT NUMBER WILL NOT BE LABELED
   • When ON, the element labels are superimposed on their respective elements.



The elements displayed with their element numbers.

6.3.10 MIRROR

Description: This function allows the user to reflect a copy of desired elements across a local or global coordinate plane.

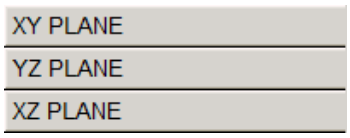
Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. Once the desired elements have been selected, VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   • VPG asks for a desired coordinate system (local or global). VPG prompts the user to create a local system.

3. Once the desired coordinate system is acquired, it will be displayed on the screen, and the prompt will read:
   > ACCEPT. (Y/N/A)
   • If the user selects NO, VPG returns to step 2.

4. If the user selects YES, VPG prompts:
   > ENTER PLANE OF MIRROR (XY, YZ, XZ)



5. VPG prompts whether or not to keep the elements in their original parts:
   > INCLUDE MIRRORED ELEMENTS IN ITS ORIGINAL PART? (Y/N/A)
   ? YES/ NO
   - **YES** will place all selected elements into their source part.
   - **NO** will place all selected elements into the CURRENT PART.

6.3.11 MODIFY

Description: This function allows the user to simultaneously delete and recreate any displayed elements. The displayed elements that are to be modified DO NOT need to be in the CURRENT part. VPG will automatically identify the element type of the selected element and prompt the user to recreate it.

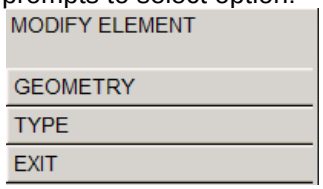
Details are covered in CREATE ELEMENTS earlier in this section.

In LS-DYNA mode, MODIFY allows the user to edit element attributes (i.e. eid, pid, nid), if the ATTRIBUTE TABLE ON / OFF function in the ELEMENT OPTIONS menu is on. For further description, see the command ATTRIBUTE TABLE ON / OFF in this section.

Note: For one-dimensional elements (beams, bars, etc.), VPG displays the selected beam's property ID and allows the user to modify it and the beam's connectivity.

Usage:

1. VPG prompts to select option:



   - If GEOMETRY is selected, go to step 2, if TYPE is selected, go to step 3.

2. If GEOMETRY is selected, VPG then prompts:
   > PICK ELEMENT TO MODIFY
   - select an element, VPG then guide user to select nodes to regenerate element

3. If TYPE is selected, user has following options to change element type
   - **PLOTEL TO BEAM.**, change selected PLOTEL element to beam element.
   - **ACM3 TO MAT 100 SP.**, changes all ACM3 link in database to mat 100 beam spot weld.
   - **DIS. TO MAT 100 SP.**, changes all discrete beam link in database to mat 100 beam spot weld.
   - **RBE2 TO SPRING**, changes selected RBE2 link in database to spring.
   - **SPOTWELD TO MAT 100 SP.**, changes all spot weld link in database to mat 100 spot weld.
   - **SPOTWELD TO RBE2.**, changes selected spot weld link in database to rbe2.
   - **SPRING TO SPOTWELD**, changes selected spring in database to spot weld.
   - **DELETE SHELL THICKNESS**, deletes the thickness of selected shells.

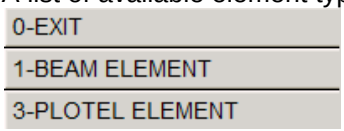
- **MAT100 BEAM TO BLOCK**. changes selected mat100 beam spot weld to mat100 block spot weld.

6.3.12 LINE MESH

Description: This function allows the user to generate one-dimensional elements along a selected line automatically. The elements created will follow the direction of the selected line. The node sequence of the element begins at the starting point of the selected line according to the right hand rule. The created elements will be included in the CURRENT PART. (The CURRENT PART should be verified before proceeding with this function.)

Usage:

1. VPG prompts:
   > SELECT ELEMENT TYPE
   - A list of available element types will be displayed, as following:



2. VPG then prompts:
   > SELECT LINE FOR LINE MESH

3. Once the line selection has been made, VPG prompts:
   > ENTER NUMBER OF ELEMENTS ON SELECTED LINE
   - Any integer greater than 1 is valid.

4. VPG then prompts for the beam orientation:
   > PICK NODE/PT OR VECTOR FOR BEAM/GAP ORIENTATION

5. VPG prompts:
   > ACCEPT MESH (YES/NO)
   ? YES/NO
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.

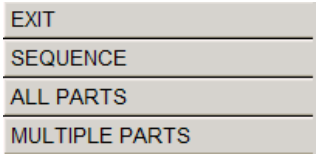
6. ABORT, DONE, or EXIT exits this function.

6.3.13 RENUMBER

Description: This function allows the user to renumber all elements in the existing database with the option of creating a summary table report of the element ranges between each part. The user may renumber elements in sequence or by parts.

Usage:

1. VPG prompts:
   > ENTER RENUMBERING OPTION:



2. When user enters **SEQUENCE,** VPG prompts:
   > ENTER STARTING ELEMENT NUMBER (DEFAULT = 1)

3. When user enters **ALL PARTS,** VPG prompts:
   > ASSIGN STARTING ELEMENT NUMBER FOR EACH PART? (Y/N)
   ? YES
   > ENTER STARTING ELEMENT NO. FOR PART: xxxx.
   • Once a number has been entered, the prompt will read;
   > ELEMENT x TO xx ASSIGNED, NEXT ELEM NO.: yyyy
   • If "blank" or "zero" is entered, yyyy will be used as the starting element number for the next part.
   • These prompts will continue for every part in the database.
   NO
   > ENTER STARTING ELEMENT NUMBER (any positive integer)
   > ENTER ELEMENT INCREMENT BETWEEN PARTS (1000)
   • Any positive integer may be entered.

4. When user enters **MULTIPLE PARTS,** VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART
   > ENTER STARTING ELEMENT NUMBER FOR CHOSEN PARTS:

6.3.14 REVERSE NORMAL

Description: This function allows the user to reverse the element orientation (normal) for all elements except CORD2R and CONM2.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. Once desired elements have been selected, DONE executes the command.

6.3.15 SPLIT

Description: This function divides CQUAD4 and CTRI3 elements into 2 or 4 elements.

Usage:

1. VPG prompts:
   > SELECT OPTION FOR SPLIT ELEMENT

2 QUADS
4 QUADS
2 TRIAS
TRIA 2 TRIAS
TRIA QUA COMBINE
TRIA 4 TRIAS
ARBITRARY LOCATION
SOLID TO 8 SOLIDS
1 DIMENSION ELEMENT
ALONG LINE
EXIT

? "**2 QUADS**" splits a QUAD4 into two QUAD4s.

"**4 QUADS**" splits a QUAD4 into four QUAD4s.
"**2 TRIAS**" splits a QUAD4 into two TRIAs.
"**TRIA 2 TRIAS**" splits a TRIA3 into two TRIA3s.
"**TRIA QUAD COMBINE**" splits a QUAD into a QUAD and a TRIA.
"**TRIA 4 TRIAS**" splits a TRIA into four TRIAs.
"**ARBITRARY LOCATION**" defines locators for splitting elements.
"**SOLID TO 8 SOLIDS**" splits a SOLID into 8 SOLIDs.
"1 DIMENSION ELEMENT" splits a 1 dimension element to several 1 dimension elements
"ALONG LINE" split shell elements by line.

2. VPG prompts:

> SELECT ELEMENT TO SPLIT

CURSOR LOCATION
MULTI-POINT REGION
BY PART
REVERSE OPERATION
DRAG WINDOW
FREE REGION
SPREAD SHELL
CIRCULAR REGION
UNDO LAST
DONE
EXIT

? **REVERSE OPERATION**

- Reverses the split direction. This option does not apply to group operations.

6.3.16 SURFACE MESH

Description: This function allows the user to automatically mesh surface data. VPG automeshes using 2D elements (plate elements). Supported surface types are covered in Pre Processor, Section 2.2, of this manual. The user has the option of automeshing by selecting one surface, multiple surfaces, or all of the displayed surfaces.

DISPLAYED SURFACES allows the user to automesh all surfaces at once. To model one surface at a time (default), the user selects an individual surface with the cursor and selects DONE.

Usage:

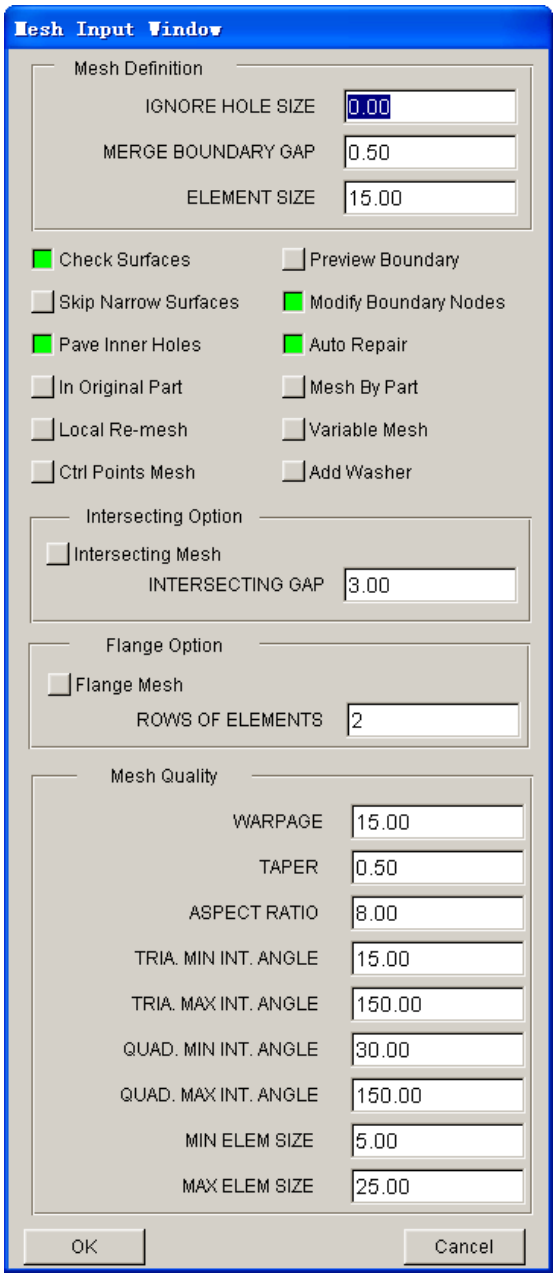
1. VPG prompts to select surface mesh methods. Surface mesh methods as following:

TOPOLOGY MESH
TRIANGULAR MESH
EXIT

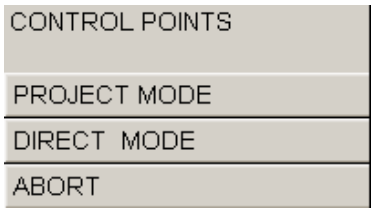
Enter **TOPOLOGY MESH**

2. VPG prompts to select surface.

3. After selecting surface, the "Mesh Input Window" pops up and allows parameters to be set in the window. Click OK to close the window.

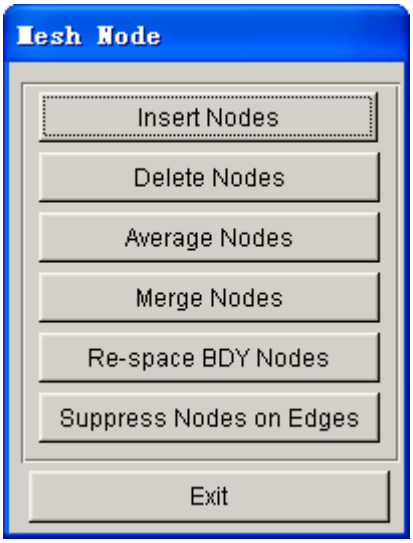


Check Surfaces: check and find duplicate surfaces.
Skip Narrow Surfaces: ignore narrow surfaces during mesh.
Pave Inner Holes: generate a layer of quad elements around holes during mesh.
In Original Part: generated elements will be put in its original part, if this option is off, those elements will be put in current part.
Local Re-mesh: meshed surface will be remeshed and boundary nodes will connect to elements generated by new meshed surfaces.
Ctrl Points Mesh: select points or nodes as control points to generate elements during mesh.



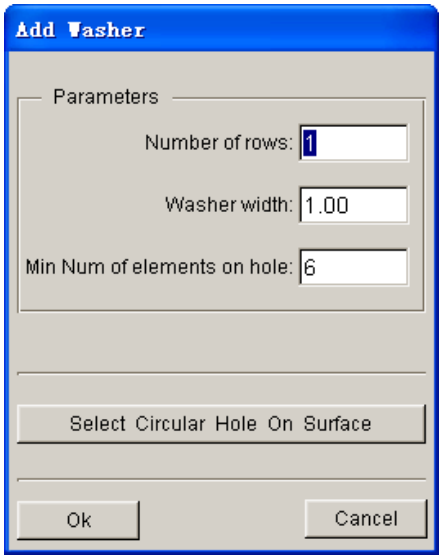
Project Mode: selected control points/nodes will be projected to selected surface to generate nodes.
Direct Mode: selected control points/nodes will directly generate nodes at their original position.

Preview Boundary: preview the boundary nodes before generating elements.
Modify Boundary Nodes: will allow user to modify boundary nodes, see following figure



Insert Nodes: insert nodes between boundary nodes.
Delete Nodes: delete selected boundary nodes.
Average Nodes: generate a node at the middle position of two selected adjacent nodes
Merge Nodes: merge two selected close nodes, not merge adjacent nodes.
Re-space BDY Nodes: respace boundary nodes for equal distance
Suppress Nodes on Edge: select common edge to suppress, nodes will not be generated on suppressed edge.

Auto Repair: this option helps to remove small gaps and duplicate elements during mesh.
Mesh By Part: allows to mesh part one by one.
Variable Mesh: generate variable elements according to the geometry feature.
Add Washer: generate washer elements around a hole, see following figure.



Number of rows: number of layers for the selected holes
Washer Width: total width of the washer
Min Num of elements on holes: minimum number of elements around the selected holes.
Select Circular Hole On Surface: select holes to generate washer elements.

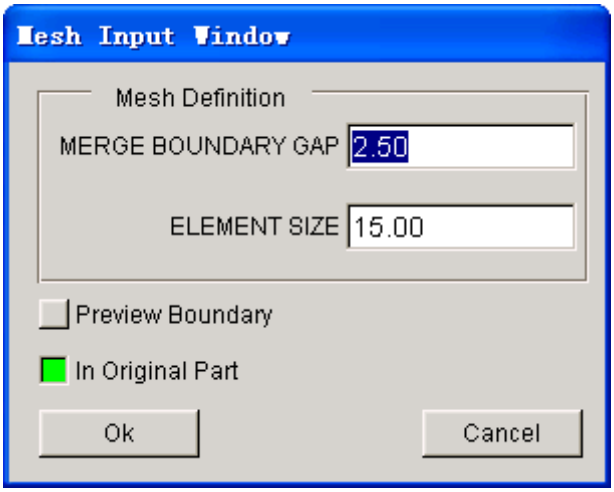
- **Intersecting Mesh**: this option helps to generate elements for intersecting surfaces, such as"T" shape, "X" shape and pipe shape surfaces. The surfaces whose mesh is not affected by other surfaces in T-Shape Type and Pipes Type can be selected as master surfaces. In X-Shape Surfaces Type, user can randomly select one surface as master surface

4. VPG prompts:
   > CONTINUE MESHING? (Y/N)
5. VPG prompts:
   > ACCEPT MESH? (YES/NO/RE-MESH)

Enter **TRIANGULAR MESH**

2. VPG prompts to select surface.

3. After selecting surface, a mesh input window is pop-up:



4. After clicking OK, VPG prompts:
   > ACCEPT MESH? (YES/NO/RE-MESH)

6.3.17 AUTO REPAIR

Description: This function allows the user to automatically repair mesh.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. After selecting elements, VPG prompts:
   > ENTER MINIMAL ELEMENT SIZE, DEFAULT IS 4.000

3. After entering the value, VPG prompts:
   > ENTER MIN INT. ANGEL, DEFAULT IS 25.00

4. After entering the value, VPG prompts:
   > ENTER MAX INT. ANGEL, DEFAULT IS 150.00

5. After entering the value, VPG prompts:
   > ENTER BOUNDARY GAP TOLERANCE, DEFAULT IS 1.000

6. Then VPG prompts:
   > ACCEPT REPAIRING RESULT?
   ? YES
   NO
   RE-REPAIRING
   - The user may re-enter above parameters and repair again.

6.3.18 2 LINE MESH

Description: This function generates elements between two selected lines. A rectangular pattern of N1 by N2 quadrilateral elements will be generated, where N1 is the number of elements along the direction of the first selected line, and N2 is the number of elements between the two lines. For a transition mesh, it is possible to enter N3 and N4 parameters for the mesh (each side may have a different number of elements e.g., 10, 5, 8, 7).

The nodes in the direction of N2 are generated by linear interpretation between the opposing nodes on the two lines. A row of N2 quadrilateral elements is generated first from the starting end of line 1 to the starting end of line 2. The pattern will be repeated N1 times along line 1. The node sequence starts at the beginning of line 1 and follows the direction of line 1 with the right hand rule. The created elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > PICK LINE
   - NODE/POINT allows the user to select node or point to create temporary lines for the 2 LINE MESH function. The user must select this option before creating the desired lines for 2 LINE MESH.
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 2 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 2 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines, the prompt will read:
   > ENTER NO. OF ELEMENTS ON EACH SIDE: N1, N2, N3, N4

   N1 - Any positive integer greater than one for the number of elements along the selected lines.

   N2 - Any positive integer for the number of elements between the 2 lines.

   N3 -(optional) an integer value for the side opposite N1 where N1 is not greater than 2*N3 and N3 is not greater than 2*N1.

   N4 -(optional) an integer value for the side opposite N2 where N2 is not greater than 2*N4 and N4 is not greater than 2*N2.
   - i.e.,10,10 or 10,10,11,14 = ACCEPTABLE
   - i.e., 5,5,12,6 = NOT ACCEPTABLE

3. Once the desired element N1, N2, N3, N4 is entered, VPG prompts:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept the mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH rejects the mesh and prompts for new N1, N2, N3, N4.

6.3.19 3 LINE MESH

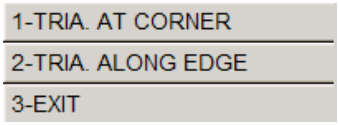
Description: This function generates elements in an area defined by 3 lines (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). Elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function. Two mesh patterns are available for 3 Line Mesh (triangular shape). VPG generates the triangular elements and allows the user to position them on the model.

Usage:

1. VPG prompts:
   > SELECT 3 LINES
   - NODE/POINT allows the user to select node or point to create temporary lines for the 3 LINE MESH function. The user must select this option before creating the desired lines for 2 LINE MESH.

- LINE SEGMENT allows the user to combine a broken line while selecting lines for the 3 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 3 LINE MESH.
- The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines, the prompt will read:
   > ENTER MESH TYPE: 1 OR 2



? 1-TRIA. AT CORNER
- Triangles are concentrated in the vertex of the first two selected lines.
  > ENTER NUMBER OF ELEMENTS: N1, N2, N3

2-TRIA. ALONG EDGE
- Triangles are concentrated along the third selected line.
  > ENTER NUMBER OF ELEMENTS: N1, N2, N3
- VPG accepts only one entry, as each side has the same number of elements.

6.3.20 4 LINE MESH

Description: This function generates elements in an area defined by 4 selected lines. A rectangular pattern of N1, N2, N3 by N4 quadrilateral elements will be generated, where N1 is the number of elements along the direction of the first selected line, and N2 is the number of elements along the direction of the second selected line, etc. For a transition mesh, it is possible to enter N3 and N4 parameters for the mesh (each side has a different number of elements e.g., 10, 5, 8, 7).

The nodes in the direction of N2 are generated by linear interpretation between the opposing nodes on the two lines. A row of N2 quadrilateral elements is generated from the starting end of line 1 to the starting end of line 2. This pattern is repeated N1 times along line 1. The node sequence starts at the beginning of line 1 and follows the direction of line 1 with the right hand rule. The created elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > SELECT LINES
   - NODE/POINT allows the user to select node or point to create temporary lines for the 4 LINE MESH function. The user must select this option before creating the desired lines for 2 LINE MESH.
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 4 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 4 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.
2. Once the user selects the desired lines, the prompt will read:
   > ENTER NO. OF ELEMENTS: N1, N2, N3, N4
   N1 - Any positive integer greater than one for the number of elements along the selected lines.
   N2 - Any positive integer for the number of elements between the 2 lines.

N3 -(optional) an integer value for the side opposite N1 where N1 is not greater than 2*N3 and N3 is not greater than 2*N1.
N4 -(optional) an integer value for the side opposite N2 where N2 is not greater than 2*N4 and N4 is not greater than 2*N2.
- i.e.,10,10 or 10,10,11,14 = ACCEPTABLE

3. Once the desired element N1, N2, N3, N4 is entered, VPG prompts:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH will not accept mesh and prompt for new N1, N2, N3, N4.

6.3.21 SOLID MESH

Description: This function opens a menu listing the following solid meshing options:

-SOLID MESH: 6 LINE SOLID MESH

Description: This function is intended for automatic 6 or 8 node solid element generation in a volume. The volume is defined by top and bottom surfaces, which are enclosed by 3 lines each. The lines must previously be defined (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). The created elements will be included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > SELECT 3 LINES AT BOTTOM
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 6 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 6 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines, the prompt will read:
   > SELECT 3 LINES AT TOP
   - Select the second set of three lines in the same order as the first set of three lines.

3. VPG prompts:
   > ENTER MESH TYPE: 1 OR 2

   1-TRIA. AT CORNER
   2-TRIA. ALONG EDGE 3

   1. > **ENTER NO. OF ELEMENTS: N1 X N2 X N3**
   2. > **ENTER NO. OF ELEMENTS ON ALONG THE FACE EDGE**
   > ENTER NO. OF ELEMENTS BETWEEN THE FACES

4. Once N1, N2, and N3 have been entered, the prompt will read:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH will not accept mesh and prompt for new N1, N2, N3.

-SOLID MESH: 8 LINE SOLID MESH

Description: This function is intended for automatic 8-node solid element generation in a volume. The volume is defined by a top and bottom surface enclosed by 4 lines. Each line must be defined beforehand (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). A row of N2 elements will be generated from the starting end of line 1 along line 4 on the bottom surface. The pattern will be repeated N1 times along line 1 on the bottom surface. The element pattern on the bottom surface will be repeated N3 times toward the top surface. The node sequence starts at the beginning of line 1 and follows the direction of line 1 using the right hand rule. The created elements will be included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

Usage:

1. VPG prompts:
   > SELECT LINES FOR BOTTOM
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 8 LINE SOLID function as in the COMBINE LINE function in the LINE menu. The user must select this option before selecting the desired lines for 8 LINE SOLID.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE menu.

2. VPG prompts:
   > SELECT LINES FOR TOP

3. Once the user selects the desired lines, the prompt will read:
   > ENTER NO. OF ELEM. N1, N2, N3
   N1 - any positive integer greater than one for the number of elements along the selected lines.
   N2 - any positive integer for the number of elements between the 2 lines.
   N3 - any positive integer for the number of elements between the bottom and top surface.

4. Once N1, N2, and N3 have been entered, the prompt will read:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH rejects the mesh and prompts for new N1, N2, N3.

-SOLID MESH: 9 LINE SOLID MESH

Description: This function allows the user to generate 6 or 8 node solid elements automatically in a volume. The volume is defined by 9 lines: 3 lines for the top surface, 3 lines for the bottom surface (see 6 LINE MESH), and 3 lines connecting the top and bottom surfaces.

Usage:

1. VPG prompts:
   > SELECT 3 LINES AT BOTTOM
   > SELECT 3 LINES AT TOP
   > SELECT 3 LINES ON THE SIDE, STARTING BETWEEN LINE 1 & 3
   - **9 LINE MESH** is similar to 6 LINE MESH with the exception of 3 connecting lines.
   - Line direction (clockwise or counter clockwise) for all line selections should be

consistent.

2. See **6 LINE MESH** for further details and the prompt descriptions.

-SOLID MESH: 12 LINE SOLID MESH

Description: This function allows the user to generate 8 node solid elements automatically in a volume. The volume is defined by 12 lines --4 lines for the top surface, 4 lines for the bottom surface (see 8 LINE MESH), and 4 lines for connecting the top and bottom surfaces.

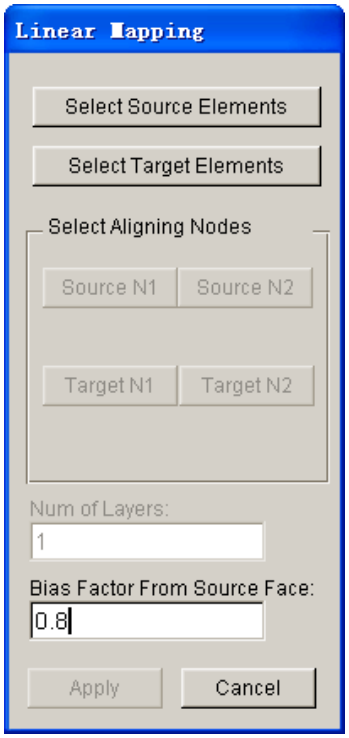
Usage:

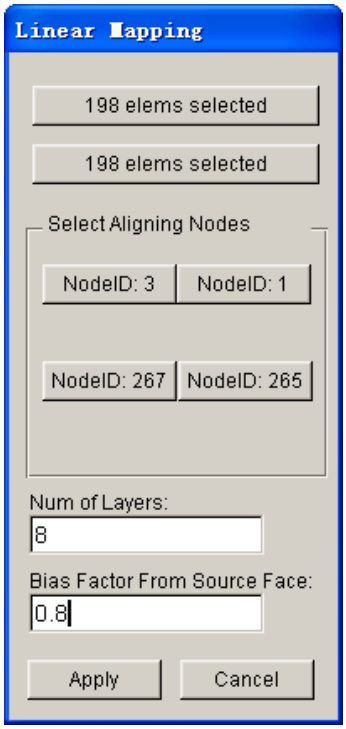
1. VPG prompts:
   - **> SELECT 4 LINES AT BOTTOM**
   - **> SELECT 4 LINES AT TOP**
   - **> SELECT 4 LINES ON THE SIDE, STARTING BETWEEN LINE 1 & 4**
   - **12 LINE MESH** is similar to **8 LINE MESH** with the exception of 4 connecting lines.
   - The user should maintain the consistent direction (clockwise, counter clock-wise) for all line selections.

2. See section **8 LINE MESH** for further details and prompt descriptions.

-LINEAR MAPPING

Description: This function allows the user to generate solid elements model by linear mapping.





Usage:

1. Select source elements and target elements, and the numbers of both elements

should be the same.

2. The distributions of node locations are similar and VPG will automatically look for the aligning nodes in the model, sometimes, user needs to manually select the aligning nodes from source and target elements.
3. Set the number of layers between source elements and target elements, and the bias factor from source face; click the Apply button to generate solid elements by linear mapping.

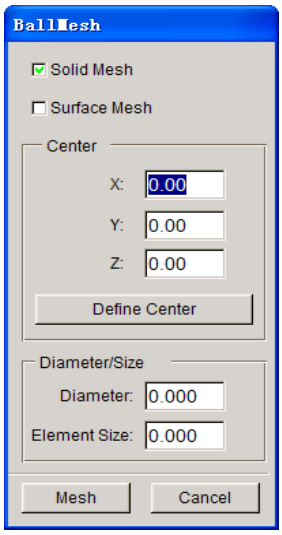
-SPHERICAL MESH

Description: The BALL MESH function creates a spherical mesh using either solid elements or shell elements. The solid element option creates a hexahedron and tetrahedron mesh of a sphere using the diameter and mesh size values input into the BALL MESH table. The sphere will be centered at the X, Y and Z location specified in the table.

If the Surface Mesh option is selected, a shell mesh will be created using the center, diameter and mesh size parameters specified.

Usage:

1. The following window pops up. The user defines parameters in the window and clicks "Mesh".

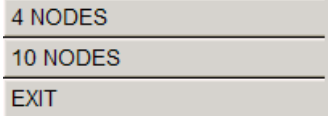


-TETRA MESH

Description: This function allows the user to generate 4 or 10 node solid elements automatically in a volume.

Usage:

1. VPG prompts:
   > SELECT TETRA MESH OPTION



2. VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART

- After parts are selected, VPG automatically generates 4/10 nodes tetra mesh.

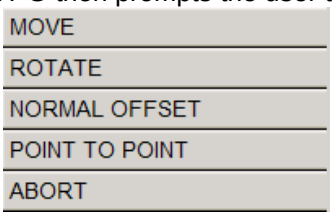
6.3.22 TRANSFORM

Description: This function allows the user to transform selected elements by move, rotate, mapping (LCS), and normal offset. The normal offset option only applies to plate elements. It should be noted that the normal of the plate elements should be consistent for a proper normal offset.

Usage:

1. VPG first prompts the user to select the desired elements.

2. VPG then prompts the user to select the desired transformation:



MOVE: moves the selected elements to a new location according to the distance entered along the selected axis.

ROTATE: rotates the selected elements to an angle increment (right-hand rule).

NORMAL OFFSET: moves the selected plate elements in the normal direction by the specified thickness.

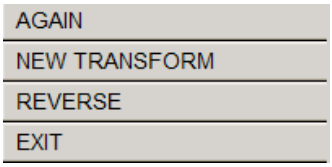
POINT TO POINT: moves the selected plate elements from one point to the other.

3. VPG prompts for a desired coordinate system (local or global) (this step is not required for NORMAL OFFSET). A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

4. VPG prompts for the transformation value.

5. The elements are transformed.

6. VPG prompts the user to select one of the following options:



AGAIN: performs the same transformation again with the same specifications.
NEW TRANSFORM: begins another transformation operation.
REVERSE: rejects the previous transformation.
EXIT: exits the function.

6.3.23 COMPLETE MESH

Description: This function allows user to fill the element holes.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS FOR FILL INNER BOUNDARY

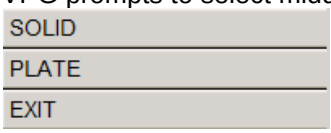
2. After selecting elements and click DONE, VPG prompts:
   > SELECT A NODE ON THE CLOSED CURVE
   - The hole is filled when the node is selected.

6.3.24 MIDDLE ELEMENT

Description: This function will create middle element between two layers of shell elements or directly by solid elements.

Usage:

1. VPG prompts to select middle element options.



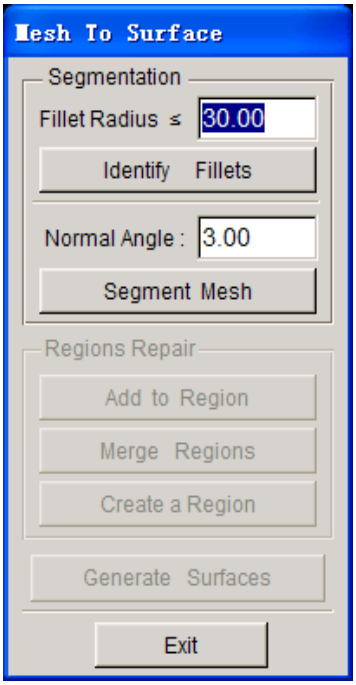
- **SOLID**, middle element will be created between two segments of solid.
- **PLATE**, middle element will be created between two lays of shell elements.

6.3.25 ATTRIBUTE TABLE ON/OFF

Description: This function, available only in LS-DYNA mode, allows the user to add additional properties to elements that VPG does not prompt for when the elements are created (thickness, additional nodes, etc.) The ATTRIBUTE TABLE supports BEAM, DISCRETE, MASS, SEATBELT, SHELL, TSHELL, and solid elements. It does not support JOINT, NODAL RIGID BODY, PLOTEL, and SPOTWELD elements.

6.3.26 MESH TO SURFACE

Description: This function creates surfaces from shell elements.



Usage:

1. VPG prompts to identify fillets and segment mesh:

   IDENTIFY FILLETS
   - Input the largest radius for fillets to identify, value between 5~50mm. User can input new radius and click this button many times.

   SEGMENT MESH
   - Input Normal Angle between 1~30 degree, and do segment mesh. User can input new angle value and click this button many times.

2. After click SEGMENT MESH, user can adjust region, if there is no need to adjust regions, user can jump over this step:

   ADD TO REGION
   - select elements, and add them to the target region.

   MERGE REGION
   - select regions one by one, and they are merged into one region, click middle mouse to accept the merge, and right mouse to cancel the merge, right mouse again to exit merge region.

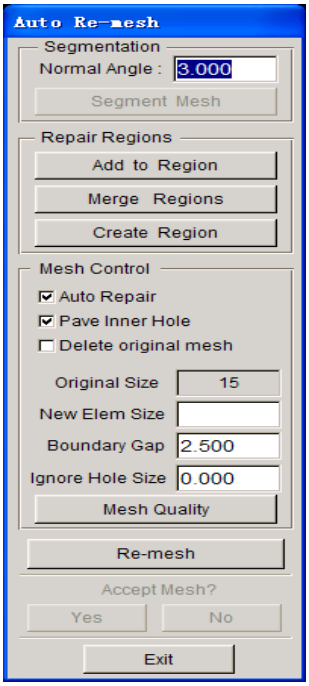
   CREATE A REGION
   - select elements to make up a new region.

3. After adjust regions, user can generate surfaces now:

   GENERATE SURFACE
   - VPG will generate surfaces according to the regions.

6.3.27 AUTO REMESH

Description: This function will automatically remesh the shell elements in database.



Usage:

1. VPG prompts to do segment mesh:

   SEGMENT MESH
   - Input Normal Angle between 1~30 degree, and do segment mesh. User can input new angle value and click this button many times.

2. after click SEGMENT MESH, user can adjust region:

   ADD TO REGION
   - select elements, and add them to the target region.

   MERGE REGION
   - select regions one by one, and they are merged into one region, click middle mouse to accept the merge, and right mouse to cancel the merge, right mouse again to exit merge region.

   CREATE A REGION
   - select elements to make up a new region.

3. User can select option for elements :

   AUTO REPAIR
   - VPG will repair elements during regenerate elements.

   PAVE INNER HOLE
   - Quadrangle elements will be meshed around the inner hole.

   DELETE ORIGINAL ELEMENTS
   - Original elements will be deleted if this option is on.

   ORIGINAL SIZE
   - VPG automatically check the original elements size.

   NEW ELEMENT SIZE
   - User needs to enter this size for regenerated elements.

   BOUNDARY GAP
   - Gaps less than this value will be merged during remesh.

   IGNORE HOLE SIZE
   - Holes less than this value will be ignored during remesh.

   MESH QUALITY

- Detailed quality set will be done in the pop up window.

4. User now can remesh elements:
   RE-MESH
   - Remesh elements starts.

5. After remesh elements, VPG prompt user whether to accept remesh results
   YES
   - Accept remesh results.

   NO
   - give up remesh results and return to remesh window .

6.3.28 SEPARATE ELEMENT

Description: This function separates connected elements; the separated elements will just have nodes of common positions but not connecting to each other.

Usage:

1. VPG prompts:
   SELECT NODES
   - Select one or several nodes, and click DONE, the elements sharing the selected nodes are separated. User can verify this separation by NODE COINCIDENT CHECK function.

6.3.29 SPECIAL ELEMENT KEYWORD

Description: This function defines some element keyword cards.

Usage:

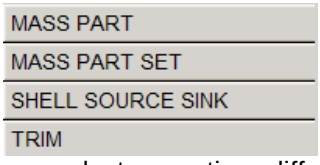
2. VPG prompts:
   CREATE
   - This function creates a card

   DELETE
   - This function deletes already created cards

   LIST
   - This function lists already created cards in pop-up window

   MODIFY
   - Allows user to modify already created cards

3. when CREATE is selected, VPG prompts:
   > SELECT OPTION

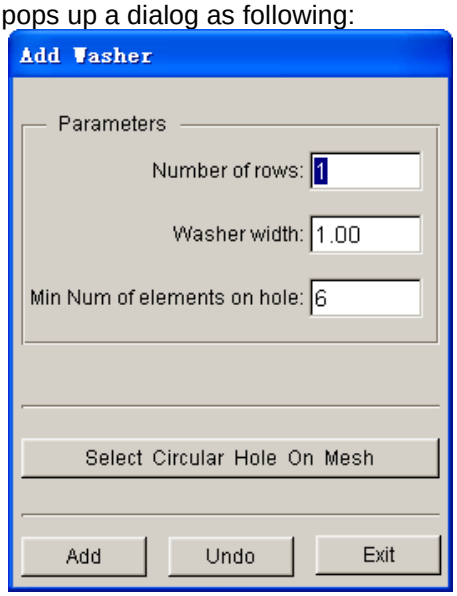
   - select an option, different cards appear, refer to LS-DYNA manual for further description.

6.3.30 ADD WASHER

Description: This function helps to remesh elements around a hole to generate washer elements.

Usage:

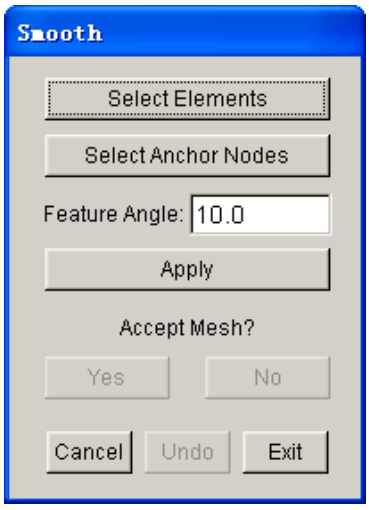
1. VPG pops up a dialog as following:



Number of rows: number of layers for the selected holes
Washer Width: total width of the washer
Min Num of elements on holes: minimum number of elements around the selected holes.
Select Circular Hole On Mesh: select holes to generate washer elements.
Add: washer elements will be generated around the hole.
Undo: reject the generated washer elements result.
Exit: exit this Add Washer function.

6.3.31 SMOOTHING

Description: This function is used to adjust the nodes of selected elements to achieve good element quality.



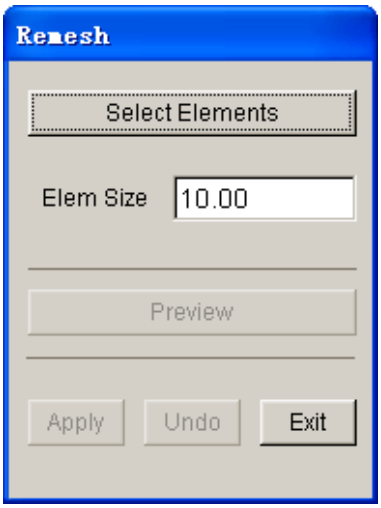
Select Elements: select elements for smoothing, selected elements are shown highlighted.
Select Anchor Nodes: select nodes which will keep fixed during smoothing.

Feature Angle: angle between element normal, as a standard to regenerate temporary surfaces during smoothing.
Apply: executing smoothing mesh for selected elements. New elements are shown highlighted, and old elements keep their original color.
Yes: accept the smoothing mesh result.
No: reject the smoothing mesh result.
Cancel: abort the elements selected for smoothing.
Undo: after clicking "Yes", user still can reject the smoothing mesh and return to the result before smoothing.
Exit: exit this smoothing mesh function.

Note:
1. The "Feature angle" option will be inactive if there is surface information in database.
2. Nodes on the surface boundary will keep unmoved during smoothing.
3. There is no matter whether to select elements or anchor nodes first when doing smoothing mesh, and before clicking "Apply", user can select elements and anchor nodes any times.

6.3.32 LOCAL REMESH

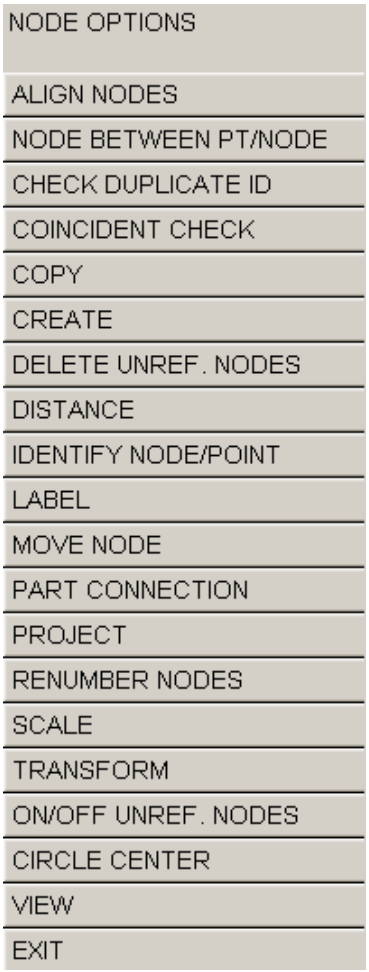
Description: Local Remesh function helps to remesh the elements region user selected. It has following features: 1.support to remesh on elements attached no-surface, 2. support to remesh elements crossing multiple surfaces; 3.support to change element size during remesh.



Select Elements: select elements region to remesh,
Elem Size: VPG gives a default element size according select elements, user can modify it as needed.
Preview: see result of local re-mesh, select elements are highlighted and new generated elements are shown as part color.
Apply: accept the remesh result, and the remesh region boundary is shown highlighted.
Undo: reject the remesh result, and the remesh region boundary is shown highlighted.
Exit: exit this local remesh function.

6.4 NODE OPTIONS

The functions in the NODE OPTIONS menu relate to the two types of nodes that are defined in the VPG program. The first type is the referenced node (connected to elements). The second type, the unreferenced node (not connected to any element), is represented with an asterisk (*). Useful applications in this section include: moving nodes (node to node or node to point), checking for and merging common or coincident nodes, and transforming or rotating nodes.



A detailed description of each function is given in the following section.

6.4.1 ALIGN NODES

Description: This function allows users to align the selected nodes along the selected lines. Nodes that have no projection to the selected line can also be aligned due to the temporary extent line of the selected line.

Usage:

1. VPG prompts:
   > SELECT A LINE TO ALIGN NODES

2. After selecting a line, VPG prompts:

> **SELECT NODES TO ALIGN**
- The selected nodes will be aligned along the selected line.

6.4.2 NODES BETWEEN PT/NODE

Description: This command generates nodes that are spaced equally between two existing selected nodes or points. The new nodes are displayed as unreferenced (free) nodes and are labeled with asterisks (*).

Usage:

1. VPG prompts for each point/node:
   > **SELECT NODE/POINT 1**

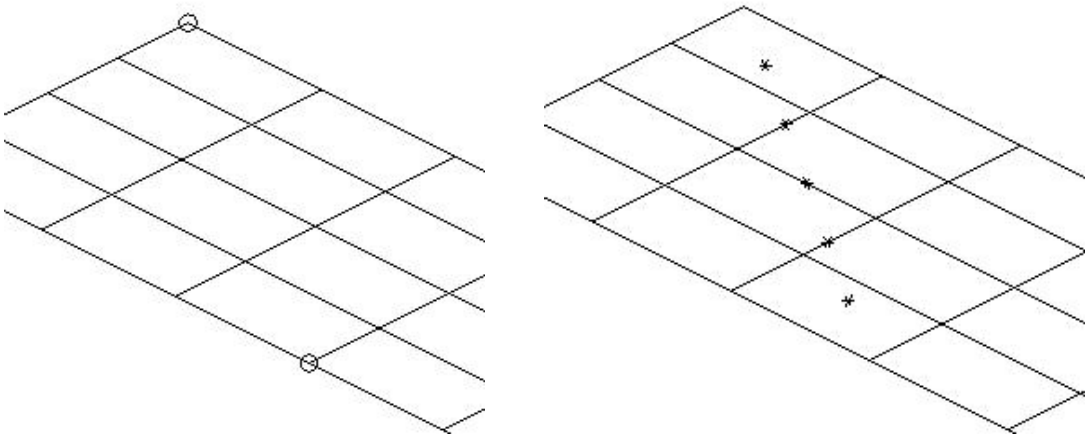
   NODE
   POINT
   REJECT LAST
   EXIT

   - **EXIT** returns the user to the NODE OPTIONS menu.
   - The user can select the node nearest the cursor by pressing the left mouse button instead of typing the letter (N).
   - VPG prompts:

   > **SELECT NODE/POINT 2**

2. VPG prompts:
   > **ENTER NUMBER OF NODES BETWEEN TWO NODES/POINTS**
   - Any positive integer is valid.

3. The user has the option to enter EXIT to terminate.



Asterisks represent unreferenced nodes that have been created between two existing nodes.

6.4.3 CHECK DUPLICATE ID

Description: This function renumbers any duplicate node numbers found in the VPG database. Users are unable to create duplicate node numbers from within the program. However, users may import files containing duplicate nodes from other programs. VPG will renumber the duplicates according to the value of the "maximum node number + 1."

Usage:

1. If any duplicate nodes are found, the following messages echo on screen and in the VPG.msg user files located in the work directory:
   > CHECKING xxxx NODES FOR DUPLICATE NODES
   > DUPLICATE NODE xxxx CHANGED TO yyyy
   > xxxx DUPLICATE NODES FOUND

   If no duplicate nodes are found, the following messages echo on screen and in the VPG.msg user files located in the work directory:
   > CHECKING xxxx NODES FOR DUPLICATE NODES
   > NO DUPLICATE NODE FOUND

6.4.4 CHANGE ID

Description: This function changes the numbers of the existing nodes. The parts containing these nodes do not need to be on or current during this procedure.

Usage:

1. VPG prompts:
   > SELECT NODE FOR NEW NODE NUMBER
   • The user can select the node nearest the cursor by pressing the left mouse button instead of typing the letter "N" or entering the keyboard option.

2. After selecting a node, the user is prompted for a new node number:
   > ENTER NEW NODE NO. FOR XXXX OR [CR] FOR UNCHANGE
   • Any unused positive integer is a valid entry. If the entered number is already being used, the following prompt will appear.
   > NODE NUMBER xxxx ALREADY EXISTS, REQUEST DENIED
   > ENTER NEW NODE NO. FOR XXXX OR [CR] FOR UNCHANGE

6.4.5 COINCIDENT CHECK

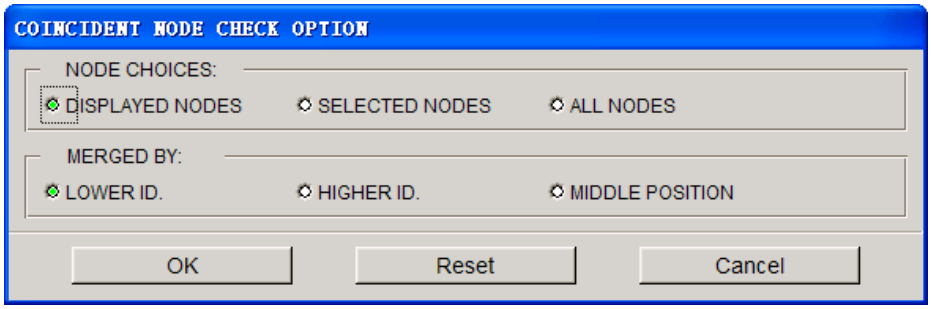
Description: This option checks for coincident nodes that may be in the displayed or other part(s) of the model. Coincident nodes are defined as having distance between each other that is less than or equal to a specified tolerance. The VPG default tolerance, which is the lowest tolerance VPG allows, is 0.01 units. The user may increase the default tolerance as necessary.

The COINCIDENT NODE CHECK merges two nodes in a coincident node group into one node and retains the lower node number (i.e., VPG combines two nodes that share the same location and renumbers the new node with the lower grid number). VPG designates the higher grid number as unreferenced.

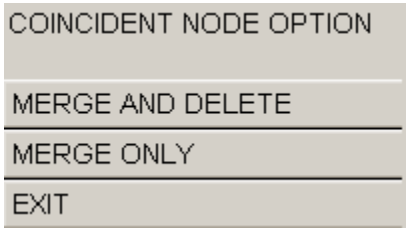
Usage:

1. VPG prompts:
   > ENTER TOL (DEFAULT=0.01)
   • Any positive real number is a valid entry.

2. After entering a tolerance, VPG prompts the user to select the nodes to be checked:
   > SELECT OPTION FOR COINCIDENT NODE CHECK



- **Cancel** Exits the user to the NODE OPTIONS menu without executing COINCIDENT NODE CHECK.
- After selecting one of the previous options, or if no coincident nodes are found, the user returns to the NODE OPTIONS menu.
- If coincident nodes are found, VPG prompts:
  - **> XXXX COINCIDENT NODES FOUND**
  - **> PICK AN OPTION TO OPERATE FOUND COINCIDENT NODES**



- **MERGE AND DELETE**: Unreferenced nodes after merging will be deleted
- **MERGE ONLY**: Coincident nodes become unreferenced nodes and are represented by asterisks (*).
- VPG will not merge nodes belonging to the same element (i.e. zero length elements with unique node numbers will not be merged, avoiding the degeneration of the element).

6.4.6 COPY

Description: This function generates a new set of unreferenced nodes at a user-specified location from either referenced or unreferenced nodes.

Usage:

1. VPG prompts:
   - **> SELECT NODES**
   - Once the nodes have been selected, DONE concludes this step.

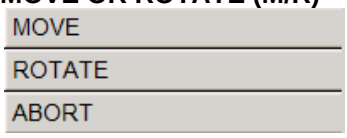
2. VPG prompts:
   - **> CREATE LOCAL COORDINATES**
   - **> PICK NODE/PT FOR ORIGIN**
   - VPG prompts the user to create a local coordinate system.

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   - **> ACCEPT? (Y/N/A)**
   - If YES, the user goes to step 4. If NO, the user returns to step 2. If ABORT, the user exits the function COPY and returns to the NODE OPTIONS menu.

4. If the user selects YES, VPG prompts for the number of copies and the type of transformation:

> ENTER NUMBER OF COPIES (positive integer)

> MOVE OR ROTATE (M/R)



? MOVE

> ENTER U, V, W INCREMENTS

- e.g. 0,0,1000. This entry will copy the selected node(s) 1000 units along the W-axis.

ROTATE

> ENTER ANGLE INCREMENT

- Enter the angle increment (right hand rule) for local W or global Z.

ABORT

6.4.7 CREATE (NODE)

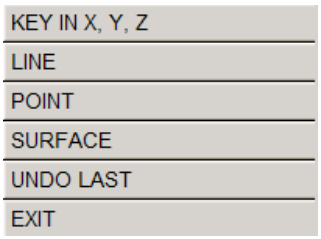
Description: This function generates UNREFERENCED NODES on points or by entering the coordinates etc.

Usage:

1. VPG prompts:

> SELECT METHOD TO CREATE NODE (POINT IS DEFAULT)

2. The user may now select node locations by:



- **KEY IN X, Y, Z**, enter the coordinate of the node to be created.
- **LINE**, select a line, create node by cursor or by entering the number of nodes on this line.
- **POINT**, create a node at the selected point.
- **SURFACE**, create nodes by cursor on the selected surface.
- **UNDO LAST**, remove the last created node.
- **EXIT**, the user will return to the NODE OPTIONS menu.

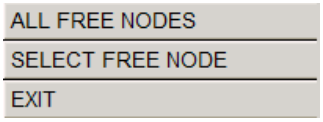
6.4.8 DELETE UNREF. NODES

Description: This function allows the user to DELETE either ALL or SELECTED unreferenced nodes in a database. Unreferenced nodes are labeled with asterisks (*).

Usage:

1. VPG prompts:

> DELETE ALL OR SELECT FREE NODES? (A/S/E)



? ALL FREE NODES

- Deletes all unreferenced nodes.

SELECT FREE NODE

- The user may now select individual or grouped nodes.
- If any of the selected nodes are referenced nodes, VPG displays the message:
  > CANNOT DELETE NODE xxxx, IT IS NOT UNREF.
- When unreferenced nodes are deleted, VPG will echo the following message:
  > xxxx unreferenced nodes deleted

6.4.9 DISTANCE

Description: This function calculates the distance between two points, two nodes, or a node and point. The calculated distance will be displayed in the Message area.

Usage:

1. VPG prompts:
   > SELECT FIRST NODE/POINT FOR DISTANCE

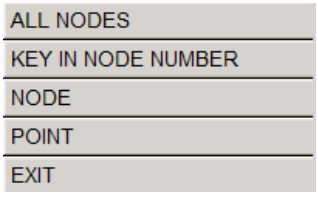
2. VPG prompts:
   > SELECT A NODE/POINT FOR DISTANCE
   - The overall distance and the global DX, DY, DZ components will be displayed at the prompt area of the screen.
   - The user enters ABORT, DONE, or EXIT.

6.4.10 IDENTIFY NODE/POINT

Description: This function identifies any point/node and its corresponding global location in terms of its X, Y, Z coordinates and node or point number.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS



- If **ALL NODES** is selected, VPG displays the appropriate numbers at the node locations.
- If **KEY IN NODE NUMBER** is selected, VPG displays the appropriate node location. The following message will echo on the command line:
  > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
- If **NODE** is selected, VPG displays the number of the node nearest the cursor location. The following message will echo on the command line:
  > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
- If **POINT** is selected, VPG displays the number of the point nearest the cursor

location. The following message will echo on the command line:

> POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz

- **EXIT** completes the command sequence and returns the user to the NODE OPTIONS menu.

Note: To clear the label display, select CLEAR from the VPG DISPLAY PARAMETER OPTIONS module.

6.4.11 LABEL

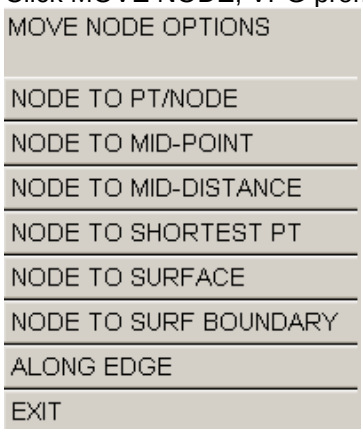
Description: This function toggles the node labels ON/OFF. When the user selects LABEL, VPG displays the numbers of the nodes in the database.

6.4.12 MOVE NODE

Description: This function moves referenced or unreferenced nodes to any location on the display screen.

Usage:

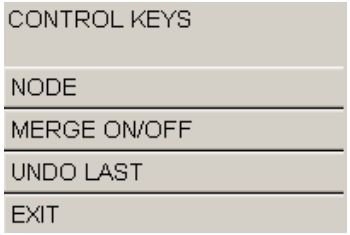
1. Click MOVE NODE, VPG prompts:



6.4.12.1 MOVE NODE TO PT/NODE

1. If NODE TO PT/NODE is selected, VPG prompts:

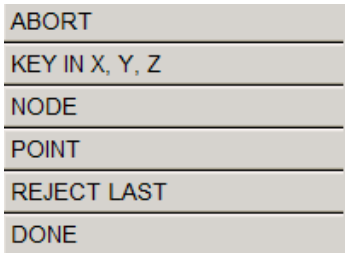
> SELECT NODE TO MOVE



- MERGE ON/OFF is an switch option for merging moved node and the target node, the default option is on, and the moved and target node will be merged.
- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.
- If a coincident node is encountered, skip to step 3 and then to step 2.

2 After selecting a node, VPG prompts for a new location for the nodes:

> PICK NEW LOCATION



- The user may move to an existing point/node by mouse or keyboard entry.
- DONE or REJECT LAST returns the user to step 1.
- ABORT exits this function and returns the user to the NODE OPTIONS menu.

3. If multiple or coincident nodes are found near the cursor, VPG prompts:
   > COINCIDENT NODES FOUND
   > SELECT ELEMENT FOR NODE
   - If an ELEMENT center is selected by the CURSOR and it is unconnected to the target node, VPG prompts:
     > SELECTED NODE IS NOT CONNECTED TO DUPL. NODE
     > SELECT ELEMENT FOR NODE
   - After this command is completed, the user returns to step 1.

6.4.12.2 MOVE NODE TO MID-POINT

1. If NODE TO MID-POINT is selected, VPG prompts:
   > SELECT NODE TO MOVE

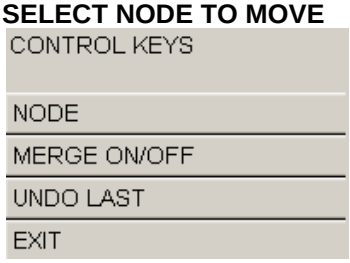
   - UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

2 After selecting a node, VPG prompts:
   > SELECT FIRST REFERENCE NODE

3. After selecting the first reference node, VPG prompts:
   > SELECT SECOND REFERENCE NODE
   - After this command is completed, the user returns to step 1.

6.4.12.3 MOVE NODE TO MID-DISTANCE

1. If NODE TO MID-DISTANCE is selected, VPG prompts:
   > SELECT FIRST REFERENCE NODE
   > SELECT NODE TO MOVE



   - MERGE ON/OFF is an switch option for merging moved node and the target node, the default option is on, and the moved and target node will be merged.

- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

2 After selecting a node, VPG prompts:
> SELECT SECOND REFERENCE NODE

3. After selecting the second reference node, VPG prompts:
> MERGE TWO MOVED NODES?
? YES
NO
ABORT
- If YES is selected, two moved nodes are merged.
- After this command is completed, the user returns to step 1.

6.4.12.4 MOVE NODE TO SHORTEST PT

1. If NODE TO SHORTEST PT is selected, VPG prompts:
> SELECT NODE TO MOVE

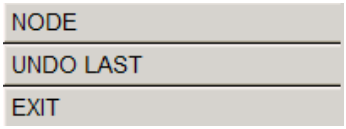
- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

2 After selecting a node, VPG prompts:
> SELECT FIRST REFERENCE NODE

3. After selecting the first reference node, VPG prompts:
> SELECT SECOND REFERENCE NODE
- After this command is completed, the user returns to step 1.

6.4.12.5 MOVE NODE TO SURFACE

1. If NODE TO SURFACE is selected, VPG prompts:
> SELECT NODE TO MOVE

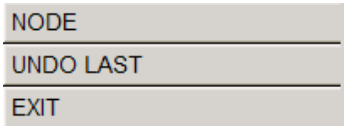


- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

2 After selecting a node, VPG prompts:
> SELECT A POINT ON SURFACE
> SELECT SURFACE
- After this command is completed, the user returns to step 1.

6.4.12.6 MOVE NODE TO SURF BOUNDARY

1. If NODE TO SURF BOUNDARY is selected, VPG prompts:
> SELECT NODE TO MOVE



- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

2 After selecting a node, VPG prompts:
> SELECT A POINT ON SURFACE BOUNDARY
> SELECT SURFACE
- After this command is completed, the user returns to step 1.

6.4.12.7 ALONG EDGE

1. If ALONG EDGE is selected, VPG prompts:
> SELECT NODE TO MOVE

- UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.

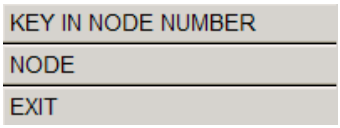
2 After selecting a node, VPG prompts:
> SELECT LOCATION ON ELEMENT EDGE
- Select a highlighted element edge, the node is moved, after this command is completed, the user returns to step 1.

6.4.13 PART CONNECTION

Description: This function determines the names and identification numbers (PID) of all of the parts that share the selected node. The message is displayed in the VPG prompt area of the screen and the nodes are highlighted with small, white circles.

Usage:

1. VPG prompts:
> SELECT NODE TO SHOW PART CONNECTION



- Once a node is selected, the part connection is listed, and the user returns to step 1.
- If the user selects a node number by KEYBOARD ENTRY that does not exist in the database, VPG prompts:
> NODE xxxx IS NOT IN DATABASE
- If the user selects a node number by KEYBOARD ENTRY that is not connected to a part, VPG prompts:
> NODE xxxx IS NOT CONNECTED TO ANY PART

6.4.14 PROJECT

Description: This function projects a node or group of nodes onto a plane, surface(s), or mesh. The nodes may be referenced or unreferenced (referenced nodes would then project the finite element mesh).

Usage:

1. VPG prompts:
   > SELECT PROJECT OPTION
   ? EXIT
   F.E. MESH (go to step 2)
   PLANE IN LOCAL UV (go to step 3)
   SURFACE (go to step 5)

2. Option (**F.E. MESH**):
   > SELECT PLATE ELEMENTS FOR TARGET SURFACE
   > SELECT ELEMENTS
   - ABORT exits without action.
   - DONE completes the operation and prompts the user to step 7.

3. Option (**PLANE IN LOCAL UV**)
   > DEFINE PROJECTION (UV) PLANE
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system to define the UV plane. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

4. VPG prompts:
   > ACCEPT? (Y/N)
   - YES go to step 9
   - NO go to step 7

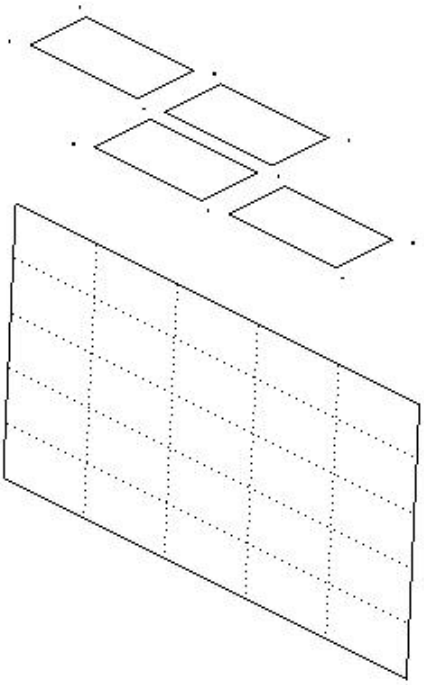
5. Option (**SURFACE**), VPG prompts:
   > SELECT SURFACE
   - ABORT exits without action. DONE completes the operation and continues to step 6.

6. VPG prompts:
   > PROJECT TO SURFACE ALONG THE NORMAL? (Y/N)
   - YES go to step 9
   - NO go to step 7

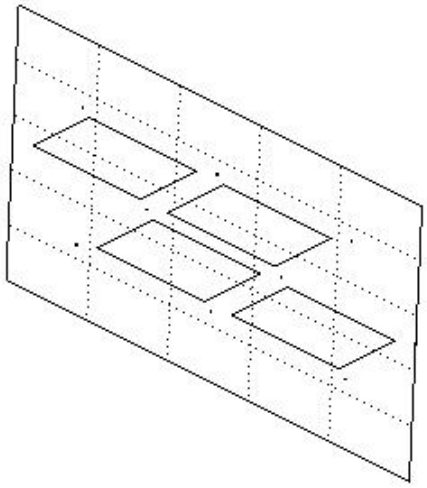
7. VPG Prompts:
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system to define the projection vector.

8. VPG displays the desired coordinate system and prompts:
   > ACCEPT? (Y/N/A)

9. If the user selects YES, VPG prompts:
   > SELECT NODE FOR PROJECTION
   - DONE or EXIT completes the operation and exits the PROJECT NODE menu.



Elements to be projected onto the surface using PROJECT NODE.



Surface with projection of elements.

6.4.15 RENUMBER NODES

Description: Renumbers all nodes in the existing database. There is also an option of creating a summary table report of the node ranges between each part. The user may renumber nodes by sequence or parts.

Usage:

1. VPG prompts:
   > ENTER RENUMBERING OPTION:

   EXIT
   ALL PARTS
   MULTIPLE PARTS

2. **ALL PARTS** go to step 4.

3. **MULTIPLE PARTS** go to step 5.

4. VPG prompts for a starting node number:
   > ASSIGN STARTING NODE NUMBER FOR EACH PART? (Y/N)
   ? YES
   > ENTER STARTING NODE NUMBER FOR PART "xxxx"
   - Once a number has been entered the prompt will read:

   > NODE x TO xx ASSIGNED, NEXT NODE NO.: xxxx
   > ENTER STARTING NODE NUMBER FOR PART "xxxx"
   - These prompts will continue for every part in the database.
   - If the user enters a STARTING NUMBER that already exists, VPG

gives the message that the location has already been assigned and returns the user to the beginning of this step.

> RENUMBER SUMMARY IS WRITTEN ON FILE *.ntb

- Zero (0) will assign (MAX NODE NO. + 1).

NO

> ENTER STARTING NODE NUMBER, -1 TO EXIT

- Once a number has been entered, the prompt will read:

> ENTER NODE NUMBER INC. BETWEEN PARTS:
> 0 – DEFAULT TO 1000, -1 – NO GAP BETWEEN PARTS

- Any integer may be entered.

> RENUMBER SUMMARY IS WRITTEN ON FILE *.ntb

5. VPG prompts:

> PICK AN ELEMENT OR PART NAME OF A PART

- After parts are selected, VPG prompts:

> ENTER STARTING NODE NUMBER FOR CHOSEN PART:

- Any integer may be entered.

6.4.16 SCALE

Description: This section applies a scale factor to any or all X, Y, and Z components for some or all nodes in a global or local direction. The scale factor may be positive or negative (change the sign on any or all of the X, Y, Z components). The parts will be magnified if the scale factor is greater than 1 and reduced if the scale factor is less than 1. The user may reflect and/or enlarge or reduce a part without retaining a copy of the original grids/elements.

Usage:

1. VPG prompts:

> SELECT NODES

- Once the user selects the desired nodes and clicks DONE, VPG prompts:

> CREATE LOCAL COORDINATES
> PICK NODE/PT FOR ORIGIN

- VPG prompts the user to create a local coordinate system.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:

> ENTER X, Y, Z SCALE FACTORS

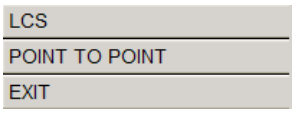
- All real numbers are valid. ZEROS or BLANKS will default to (+1).
  EXAMPLES:
  - 1, -1, 1 will reflect the selected nodes across the XZ plane (Y-direction). All Y components will be multiplied by (-1).
- -2, 1, 0.5 will magnify the X components by two times their size and reflect the nodes across the YZ plane (X-direction). These coordinates will not affect the Y components, but they will shrink the Z-components by half their size and reflect the nodes on the original side of the XY plane.
- **DONE** returns the user to the **NODE OPTIONS** menu.

6.4.17 TRANSFORM

Description: This function translates or rotates the selected nodes to new locations.

Usage:

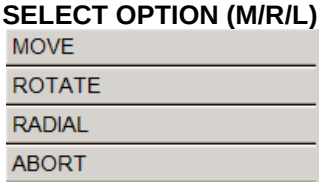
1. Click TRANSFORM, VPG prompts:



- If click LCS, go to step 2, if click POINT TO POINT, go to step 5.

2. If click LCS, VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system.
   - Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
     > ACCEPT? (Y/N/A)

3. If NO, the user returns to step 2. If YES, VPG prompts:
   > SELECT OPTION (M/R/L)



   ? MOVE
   > ENTER U, V, W INCREMENTS
   - e.g., 0,0,1000. This will move/transform the selected node(s) 1000 units along the W-axis.

   ROTATE
   > ENTER ANGLE INCREMENT (RIGHT HAND RULE**)**
   ABOUT LOCAL W OR GLOBAL Z

   RADIAL
   > ENTER INCREMENTAL RADIUS

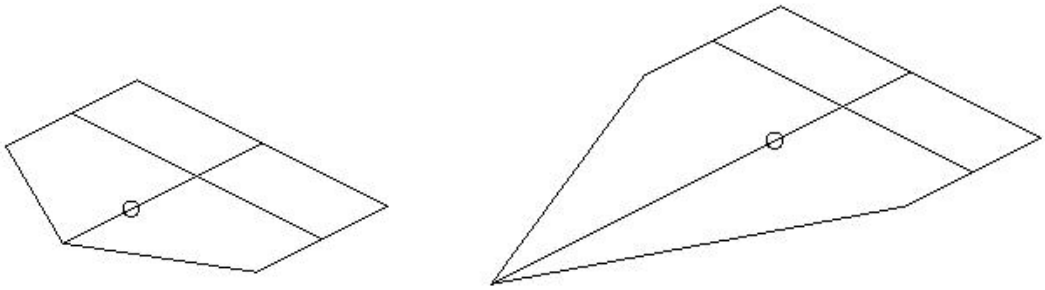
   ABORT

4. VPG prompts:
   > SELECT OPTIONS TO SELECT NODES
   - After the nodes have been chosen, VPG translates them to their new location. Selecting again will repeat the translation. Clicks EXIT to return to step 1.

5. VPG prompts:
   > SELECT NODES
   - After the nodes have been chosen, VPG translates them to their new location. Selecting again will repeat the translation.

6. After selecting nodes and clicks DONE, VPG prompts:
   > SELECT REFERENCE LOCATION

7. After selecting a node, VPG prompts:
   > SELECT TARGET LOCATION
   - Select a target node, then the nodes selected in step 5 are transformed..



6.4.18 ON/OFF UNREF. NODES

Description: This function toggles unreferenced nodes ON/OFF for viewing purposes.

6.4.19 CIRCLE CENTER

Description: This function creates a node at the center of three nodes or points.

Usage:

1. Click CIRCLE CENTER, VPG prompts:

- **NODE**: select a node, this is the default option.
- **POINT**: select a point
- **REJECT LAST**: reject the last selected node or point.

Once three nodes or points are selected, a node is created at the center of the circle formed by these three nodes (points) automatically.

6.5 MODEL CHECKER

The functions in the MODEL CHECKER menu validate the VPG models. The values for model validation are set with defaults in VPG, but the values may be adjusted to suit the user's needs. Element orientation, size, skew, connectivity, and interior angles may be compared to the specified criteria. Elements can be checked individually or for the entire model. After a model check function is performed, an interactive inquiry function allows the user to pick individual elements for the calculated quality.

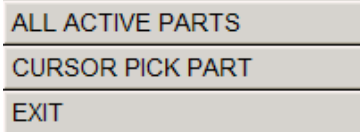
A detailed description of each function is given in the following sections.

6.5.1 AUTO NORMAL

Description: This function sets the direction of the displayed plate element normal to a selected reference plate element. The element normal is calculated by using a right hand rule as applied to the element numbering (node 1, node 2, node 3, etc.) to establish the direction of the normal.

Usage:

1. VPG prompts:
   > SELECT PART FOR AUTO NORMAL



2. User will be prompted to select a part or an element for a reference normal. The selected elements will display their current normal direction. VPG prompts:
   > IS NORMAL DIRECTION ACCEPTABLE? (Y/N)
   ? YES
   - All selected element normals will be made consistent with a referenced element normal. If normals are reversed for some elements VPG displays:
     > NORMAL REVERSED FOR XXXX ELEMENTS

   NO
   - All displayed element normals will be reversed according to the referenced element normal.

Note: Selecting one part at a time is recommended. Parts that branch off or are separated do not always get adjusted properly.

6.5.2 AUTO ORIENT SOLID

Description: This function identifies, displays, and adjusts any solid elements (hexa or tetra) which, due to element connectivity, have a non-positive volume to a positive volume.

Usage:

1. **AUTO ORIENT SOLID**
   - When the user selects AUTO ORIENT SOLID, no selection options are provided. The program automatically adjusts the display of solid element normals to a positive volume.
   - If elements are reversed, the following message will be returned:
     > ORIENTATION REVERSED FOR xxxx ELEMENTS
     - The elements that are reversed will highlight as they are checked off. The user is returned to the MODEL CHECKER menu.
   - If no elements are reversed, the following message will be returned:
     > ALL SOLID ELEMENTS PASS CHECK

     No changes occur, and the user is returned to the MODEL CHECKER menu.

Note: The user may reverse the element normals by executing the REVERSE NORMAL command in the ELEMENT OPTIONS menu.

6.5.3 ASPECT RATIO

Description: This function allows the user to check the aspect ratio of the displayed plate and solid elements. The aspect ratio is the ratio of the longest side to the shortest side of an

element. In VPG, the default aspect ratio is 8.0 to 1.0, but the user may adjust it as necessary.

Usage:

1. VPG prompts:
   > ENTER ASPECT RATIO (DEFAULT = 8.0)
   - Users may accept the default value of 8.0 by pressing the [Enter] key, or they may enter any real number that is a valid value for the aspect ratio.

   - VPG runs a check on the values for the aspect ratio. Any elements that exceed the desired aspect ratio are highlighted. The messages echoed to the command line are:
     > xxx ELEMENTS FAILED CHECK or
     > ALL ELEMENTS PASS CHECK

2. The following message will be displayed for the failed elements:
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   The PART DEFINITION window pops up.
   - The failed elements will be removed from their original parts and added to the NEW PART.
   - The new part will be ON and CURRENT.

   NO
   - VPG prompts the user to check the individual elements for aspect ratio via cursor pick. Select **EXIT** to exit the function.

6.5.4 BOUNDARY DISPLAY

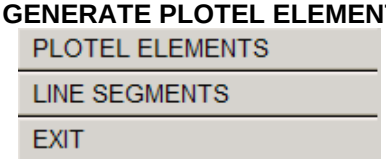
Description: This function checks the continuity of the free edges in a group of elements. All free edges will be highlighted with a boundary display.

Usage:

1. VPG prompts:
   > CHECK SINGLE OR MULTIPLE SURFACE? (S/M)
   SINGLE SURFACE
   - Checks the continuity of the displayed elements and highlights all of the single free edges of the displayed model.

   MULTIPLE SURFACE
   - Checks the continuity of the displayed elements. It also highlights the non-intersecting free edges of the multiple surfaces unless the intersection of the multiple surfaces is not connected properly.

2. VPG prompts:
   > GENERATE PLOTEL ELEMENTS FOR FREE EDGES?

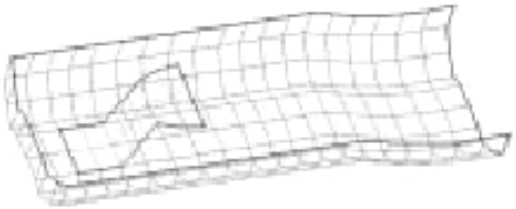
   

3. If the user selects **PLOTEL ELEMENTS**, VPG prompts:
   > INCLUDE PLOTEL ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   NO

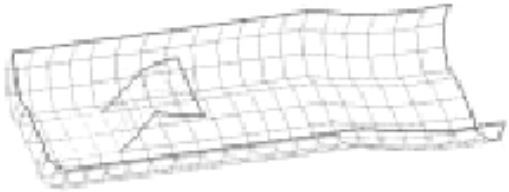
4. If the user selects **YES**, the PART DEFINITION window pops up.

- VPG will both add plotels to the part and make the part current.
- The boundary will remain highlighted until the user selects CLEAR.

Note: VPG will display the boundary of the outer most edges of a solid-element structure. Users should also use the **CHECK FREE SURFACE** command for solid element structures that are described in this section.



Single Surface Boundary Check



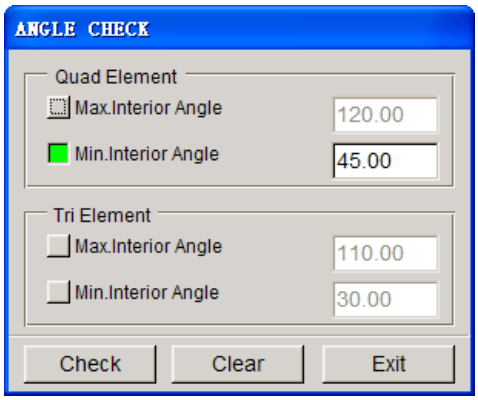
Multiple Surfaces Boundary Check

6.5.5 CHECK ANGLE

Description: This function checks the minimum values of the interior angles for shell and solid elements. VPG checks and color fills any interior angles that are less than the default values. Users have the option to adjust the default values as desired.

Usage:

1. The ANGLE CHECK window pops up and the user sees the following figure.



- Once parameters are defined, click “Check”.
- If the elements pass the interior angle check, VPG echoes the following message:
  > ALL ELEMENTS PASS CHECK!
- If any of the elements fail the model check, VPG prompts:
  > REPLACE FAILED QUADS. WITH TRI. ELEMENTS? (Y/N)
- Click YES, failed QUAD elements will be replaced by TRI. Elements, accept the replace result or not, the ANGLE CHECK dialog pops up, and VPG prompts:
  > xxxx ELEMENTS FAILED CHECK

2. If the elements fail the model check, after clicking “Exit”, VPG prompts the user to place these elements into a new part:

> INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
? YES

The PART DEFINITION window pops up.

- The failed elements will be removed from their original part and added

to the NEW PART. The new part will be ON and CURRENT.

NO

- VPG prompts the user to check individual elements for interior angle via cursor pick. Select EXIT to exit this function.

6.5.6 CHECK OVERLAP

Description: This function checks the elements for duplicate nodal connectivity. All element types are checked.

Usage:

1. CHECK OVERLAP ELEM
   - If no overlapping elements are found in the displayed elements, the following message is echoed and the user is returned to the MODEL CHECKER menu:
     > NO OVERLAP ELEMENT FOUND
   - If duplicate elements are found in the displayed elements, the following message will be echoed:
     > xxxx OVERLAP ELEMENTS FOUND

   VPG will continue with the option of adding these elements to a new part.

2. VPG prompts:
   > INCLUDE DUPLICATE ELEMENTS IN A NEW PART? (Y/N)
   ? YES

   The PART DEFINITION window pops up.
   - The duplicate elements will be removed from their original parts and added to the NEW PART. The new part will be ON and CURRENT.

   NO
   - The user will exit to the MODEL CHECKER menu.

6.5.7 CHECK JACOBIAN

Description: This function allows the user to check for element distortion against the element's ideal shape, which is given a Jacobian value of 1.0. The more an element is warped the closer it approaches a zero Jacobian value. Elements that fail the Jacobian check are color filled and the user has the option of putting them in a separate part.

Usage:

1. VPG prompts:
   > ENTER THE CRITERION FOR JACOBIAN CHECK (DEFAULT 0.7)

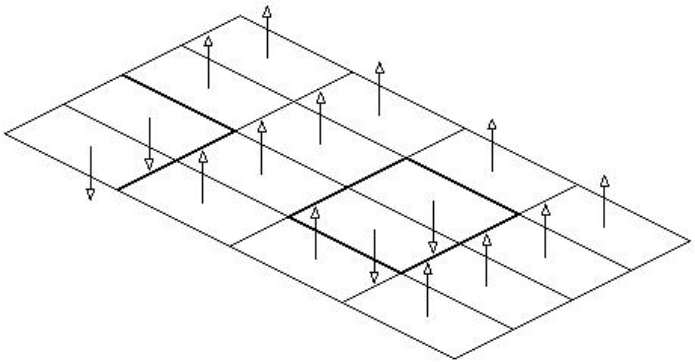
2. VPG prompts:
   > XXXX ELEMENTS FAILED CHECK
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)

3. If **YES**, VPG prompts the user to define a new part.

4. If **NO**, VPG prompts the user to check individual elements for Jacobian value via cursor pick. Select EXIT to exit the function.

6.5.8 CHECK NORMAL

Description: This function draws a white boundary line between the opposing normals of the displayed plate elements. See the following figure.



Usage:

1. **CHECK PLATE NORMAL**
   - VPG will highlight the boundary between the elements that have opposing normal orientations. Afterwards, VPG echoes the message:
     > PLATE NORMAL CHECK COMPLETED
   - The boundary will highlight when three or more plate elements connect at a common edge.
   - PLATE NORMAL in the SETUP MENU will display the element normals.

6.5.9 CHECK RIGID LINK

Description: This function checks the connectivity and length of all rigid elements, RBE2 and RBAR. Rigid bodies that exceed the length criteria of the user or that are loose, zero-length, double dependent, cyclic, or redundant are highlighted.

Usage:

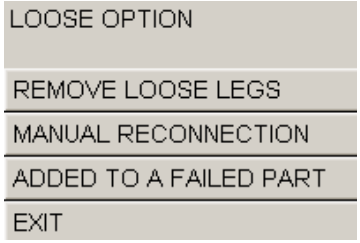
1. VPG prompts:
   > FIND RIGID LINKS LONGER THAN CERTAIN LENGTH? (Y/N)
   ? YES
   > ENTER LENGTH CRITERIA
   - Any real number greater than zero.

   NO

2. The user has the option to place failed elements into new parts. If the user opts to place the failed elements into a new part, VPG will generate the following part names:

r.xlong	• Checks for RBE2s that fail length check.
r.duplic	• Checks for degenerate RBE2s.
r.cyclic	• Checks for two RBE2s sharing the same two nodes in an independent/dependent node loop.
r.double	• Checks for RBE2s with double dependent nodes.

3. If failed elements are found, VPG prompts as following:



REMOVE LOOSE LEGS: select failed elements to remove their loose legs.
MANUAL RECONNECTION: user can manually add connection for each loose leg.
ADDED TO A FAILED PART: add failed elements to part **r.loose**
EXIT: the user returns to the MODEL CHECKER menu

6.5.10 CHECK ELEM. SIZE

Description: This function checks the minimum length of the edges of plate, solid, or beam elements. VPG color fills any elements with dimensions that are shorter than the user-defined length criteria.

Usage:

1. VPG prompts:
   > ENTER MINIMUM AND MAXIMUM LENGTH CRITERIA
   - A valid response is any real number greater than zero.
   - If the length of the of the edges of the elements is greater than the user-defined MINIMUM and is less than the user-defined MAXIMUM, then VPG echoes the message:
     > ALL ELEMENTS PASS CHECK!
   - If the length of the of the edges of the elements is less than the user- defined MINIMUM or is greater than the user-defined MAXIMUM,then VPG echoes one of the following messages:
     > ELEMENT xxxx FAILED CHECK or
     > xxxx ELEMENTS FAILED CHECK

2. If failed elements exist, then VPG prompts the user to place them into a part:
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   The PART DEFINITION window pops up.
   - The failed elements are removed from the original parts and added to the NEW PART.
   - The new part is ON and CURRENT

   NO
   > SELECT ELEMENT FOR ELEMENT SIZE
   - The user can cursor check individual elements for their size.
   - DONE or EXIT returns the user to the MODEL CHECKER menu.

6.5.11 CHECK SKEW

Description: This function allows the user to check any skew in the plate or solid elements.

Usage:

1. VPG prompts:
   > ENTER SKEW CRITERIA (DEGREE)
   - A valid response is any real number greater than zero.
   - If the skew of the elements is less than the user-defined criteria, then VPG echoes the message:
     > ALL ELEMENTS PASS CHECK!
     The user returns to the MODEL CHECKER menu.
   - If the skew of some elements is greater than the user-defined criteria, then VPG echoes one of the following messages:
     > ELEMENT xxxx FAILED CHECK or

> xxxx ELEMENTS FAILED CHECK

2. If failed elements exist, then VPG prompts the user to place them into a part:
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   The PART DEFINITION window pops up.
   - The failed elements are removed from the original parts and added to the NEW PART.
   - The new part is ON and CURRENT

   NO
   - The user returns to the MODEL CHECKER menu.

6.5.12 CHECK TAPER

Description: This function allows the user to check the length to width ratio of elements.

Usage:

1. VPG prompts:
   > ENTER THE CRITERION FOR TAPER (DEFAULT 0.5)

2. VPG prompts:
   > XXXX ELEMENTS FAILED CHECK
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   - If yes, VPG prompts the user to define a new part.

3. VPG allows the user to check individual elements for taper value via cursor pick.

6.5.13 CHECK VOLUME SKEW

Description: This function checks the shape of tetrahedral elements with respect to the deviation from a perfect pyramid shape.

Volume Skew is defined as the percentage of the volume of a tetrahedral element contains versus a sphere, containing the 4 nodes of that element.

Volume Skew = (Volume of Sphere - Volume of Tetrahedron) / (Volume of Sphere)

Usage:

1. VPG prompts:
   >ENTER VOLUE SKEW CRITERIA

2. VPG prompts:
   > XXXX ELEMENTS FAILED CHECK
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   - If yes, VPG prompts the user to define a new part.

6.5.14 CHECK WARPAGE

Description: This function color fills warpage failure elements of plate or solid. The user has the option to split the plate elements into triangular elements. All elements that fail may be added to a new part. The user may specify warpage criteria for the check function.

Usage:

1. VPG prompts:
   > ENTER CRITERIA (5 DEG)
   - Any real number is a valid entry.
   - If the warpage of the elements is less than the entered criteria, then:
     > ALL ELEMENTS PASS CHECK!
     The user is returned to the MODEL CHECKER menu.
   - If the warpage of the elements is greater than the entered criteria, then:
     > ELEMENTS xxxx FAILED CHECK
     > xxxx ELEMENTS FAILED CHECK
   - Step 3 is not applicable for solid elements. Skip to step 4 to enter the definition of the new part.

2. The user may replace warped elements:
   > REPLACE WARPED QUADS. WITH TRI. ELEMENTS? (Y/N)
   ? YES
   NO (see step 4)
   - **YES** will replace any warped quadrilateral elements with triangular elements.

3. VPG prompts:
   > SELECT ELEMENT FOR WARPAGE
   - At this point the user may pick individual elements for degree of warpage.
   - DONE or EXIT returns the user to the MODEL CHECKER menu.

An example of warped elements.

4. If some of the elements fail, and **NO** is entered, VPG prompts the user to place them into a new part:
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   The PART DEFINITION window pops up.
   - The plates and/or solids will be added to this part and the part will become current.

   NO
   > SELECT ELEMENT FOR WARPAGE
   - At this point the user may pick individual elements for degree of warpage.
   - DONE or EXIT returns the user to the MODEL CHECKER menu. The warped solids, quadrilateral elements, or newly created triangular elements will remain in their original parts.

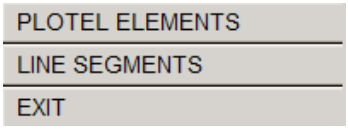
6.5.15 FEATURE LINE DISPLAY

Description: This function allows the user to check the angle between plate and solid elements based on user specified angle criteria. The user can also automatically generate plotel elements at these locations to display the structures' main features.

Usage:

1. VPG prompts:
   > ENTER FEATURE LINE ANGLE CRITERION (DEFAULT: 20 DEG)

2. VPG prompts:
   > GENERATE PLOTEL ELEMENTS OR LINES FOR FEATURE LINES?

   - **EXIT** returns the user to the model checker menu.
   - **PLOTEL (**or **LINE SEGMENTS)** prompts the following:

3. VPG prompts:
   > INCLUDE PLOTEL ELEMENTS (or **LINE SEGMENTS) IN A NEW PART? (Y/N)**
   - **NO** - Plotel elements (or Line segments) will be included in the current part.
   - **YES** - VPG prompts for a new part.

6.5.16 SECTION CUT

Description: This function cuts through the plate and solid elements at a UV-plane that the user defines and creates plotel elements or line at that plane.

Usage:

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is now asking for a coordinate system to create section plotels. VPG prompts the user to create a local system.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)
   ? YES
   > ENTER DISTANCE OF SECTION PLANE FROM W = 0
   - Any real value is a valid response.
   - User may now input a distance along the W-axis from the selected origin of the coordinate.

3. After defining coordinate system, VPG prompts:

   - User can select to generate lines or plotels.

4. If PLOTEL ELEMENTS is selected, VPG prompts:
   > INCLUDE PLOTEL ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   NO
   - If **NO** is selected, plotel elements will be added to the current part, and the user will return to the MODEL CHECKER menu.

5. If **YES** is selected, the PART DEFINITION window pops up.
   - The plotel elements will be included in the new part.
   - The new part will be ON and CURRENT.

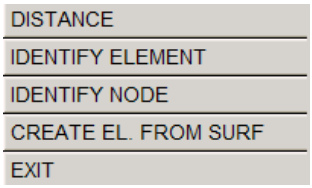
6.5.17 CHECK FREE SURFACE

Description: This function checks the continuity of the solid elements. A free surface is the face of a solid element that is occupied by one solid element. VPG will replot the solid elements on screen displaying only the free surfaces. Any unexpected quad or tri surface indicates potential errors in the solid element connectivity.

Usage:

1. CHECK FREE SURFACE
   - The following five messages will appear in the command line area of the screen:
   > XXXX SURFACES FOUND
   > CHECKING FOR FREE SURFACES
   > XXXX SURFACES CHECKED
   > CONTRUCTING FREE SURFACES
   > xxxx FREE SURFACES FOUND
   - The user is now placed in a FREE SURFACE sub-menu.

2. The following are brief descriptions of the options for the FREE SURFACE sub-menu:



DISTANCE
- This measures the distance between the points and nodes.

IDENTIFY ELEMENT
- This identifies the element numbers of the plate elements that represent the free surfaces in the FREE SURFACE sub-menu. Each plate element ID is identical to the solid element ID (i.e., one solid element with the ID xyz will have the plate elements that correspond with ID xyz).

IDENTIFY NODE
- Identifies the nodes in the solid elements and acquires the node numbers and the X, Y, Z global coordinates.

CREATE EL. FROM SURF
- Creates plate elements on the free surfaces of the displayed solids. The plate elements will reside on the screen in the current part at the time of creation.
- In order to for the user to identify any unwanted free surfaces, VPG creates plate elements that carry element/node numbers from a parent solid element. If these plate elements are used in a model, they should be renumbered with the RENUMBER ELEMENT and RENUMBER NODE commands in the ELEMENT OPTIONS and NODE OPTIONS menus, respectively.

EXIT

- Returns the user to the MODEL CHECKER menu.

6.5.18 CHECK SPRING

Description: This function checks spring elements for unconnected ends. If unconnected elements are found, a mass value can be entered to create mass elements at the free ends automatically.

Usage:

1. VPG prompts:
   > CHECK FOR NON-ZERO LENGTH SPRINGS? (Y/N)

   YES
   - This will identify all spring elements that are non-zero length and give the user the option of placing these elements in a new part.
     >XX NON-ZERO LENTH SPRINGS FOUND. INCLUDE ELEMENTS IN NEW PART? (Y/N)

2. If the user selects (Y), the elements detected will be placed in a new part named "NON-ZERO". If NO, the function is terminated.

3. VPG prompts:
   > CONTINUE TO CHECK FOR LOOSE CONNECTION? (Y/N)
   - Selecting YES will cause VPG to check for spring elements that are not connected to at least one other beam/spring/shell or solid element. Selecting NO will terminate the function.

6.5.19 CHECK JOINT

Description: This function allows the user to check for loose joint elements.

Usage:

1. VPG prompts:
   > xxxx UNCONNECTED JOINT ELEMENTS FOUND
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   NO
   - If **YES** is selected, the PART DEFINITION window pops up. After clicking OK to close the window, the user returns to the MODEL CHECKER menu.
   - If **NO** is selected, the user returns to the MODEL CHECKER menu.

6.5.20 CHECK MASS_INERTIA

Description: This function allows the user to check for loose mass and inertia.

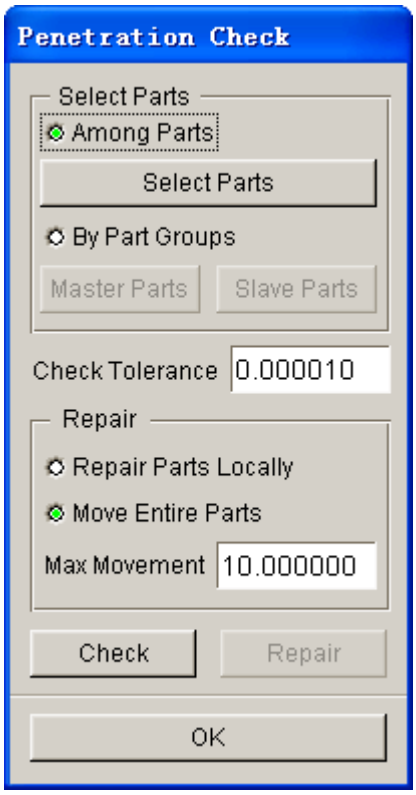
Usage:

1. If loose mass or inertia entities are found, VPG prompts:
   > XX LOOSE MASS/INERTIAS FOUND. INCLUE ELEMENTS IN NEW PART? (Y/N)

6.5.21 CHECK PENETRATION

Description: This function allows the user to check penetration. Those penetrated elements will be shown highlighted, when only two parts are selected to check penetration, VPG can also helps to repair the penetration.

Usage:



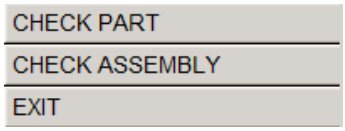
- **Among Parts**: click Select Parts button to select parts for penetration check, penetration between any two selected parts will be checked.
- **By Groups**: user needs to set the master part group and slave part group. This function is used to check the penetration between two part groups. There is no checking of penetration between parts within the part group.
- **Check Tolerance**: set tolerance value for check.
- **Repair Parts Locally**: only elements penetrated will be adjusted to remove penetration as possible.
- **Move Entire Parts**: move the whole penetrated part for a distance to remove penetration, without modifying the element shape.
- **Max Movement**: set maximum distance during one movement for node and entire part.
- **Check**: execute the penetration check function.
- **Repair**: execute the repair function.
- **OK**: accept result and exit this Penetration Check dialog

6.5.22 CHECK UNCONNECTED

Description: This function allows the user to check unconnected parts or assemblies. Unconnected parts are defined as any shell or solid mesh that is not connected via rigid elements or a node-to-node connection.

Usage:

1. VPG prompts:
   > SELECT OPTION



   - **EXIT** returns the user to the model checker menu.
   - **CHECK PART (**or **CHECK ASSEMBLY)** prompts the following:

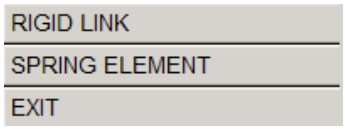
2. VPG prompts:
   > xxxx PARTS UNCONNECTED
   > UNCONNECTED PARTS TURN ON
   > CONNECTED PARTS TURN OFF
   The user returns to the MODEL CHECKER menu.

6.5.23 NORMAL ALIGNMENT

Description: This function allows the user to check normal alignment.

Usage:

1. VPG prompts:
   > SELECT THE TYPE OF ELEMENTS



   - **EXIT** returns the user to the model checker menu.
   - **RIGID LINK (**or **SPRING ELEMENT)** prompts the following:

2. VPG prompts:
   > ENTER TOLERANCE ANGLE (DEFAULT IS 8.0 DEGREES)
   - Any real number is a valid entry.
   - If there are no failed elements, VPG prompts:
     > ALL ELEMENTS PASS CHECK!
   - If there are failed elements, VPG prompts:
     > xxxx ELEMENTS FAILED CHECK
     > ADJUST THE NORMAL FOR RIGID LINK (SPRING ELEMENTS) AUTOMATICALLY? (Y/N)
     ? YES
     NO
     - If **NO** is selected, VPG prompts:
       > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
       ? YES
       NO

6.5.24 ENHANCE MESH

Description: The ENHANCE MESH function allows the user to correct modeling errors by evaluating and repairing several quality parameters simultaneously. The ENHANCE MESH function may be applied to complete models, parts, or selected elements.

The mesh quality parameters that can be repaired automatically using ENHANCE MESH are

- Element size

- Warpage
- Taper
- Aspect Ratio
- Interior Angles
- Jacobian

The user may specify the criteria that will be used to evaluate the model and will subsequently be used to attempt to improve the mesh quality.

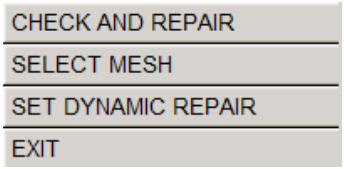
The ENHANCE MESH function uses an iterative approach to satisfy as many of the mesh quality parameters as possible. The convergence is controlled by the TOLERANCE value, and the number of iterations used to attempt to improve the mesh quality is the ITERATIONS parameter.

When the ENHANCE MESH function is executed, VPG will display a number at each element centroid, identifying the parameter(s) that have failed for that element. For instance, a value of 1,2,4 displayed at the element would indicate that the element has failed the warpage, taper, and interior angle criteria.

By selecting the FIXED BOUNDARY option, the outer boundaries of the model will not be modified during the mesh repair operations.

Usage:

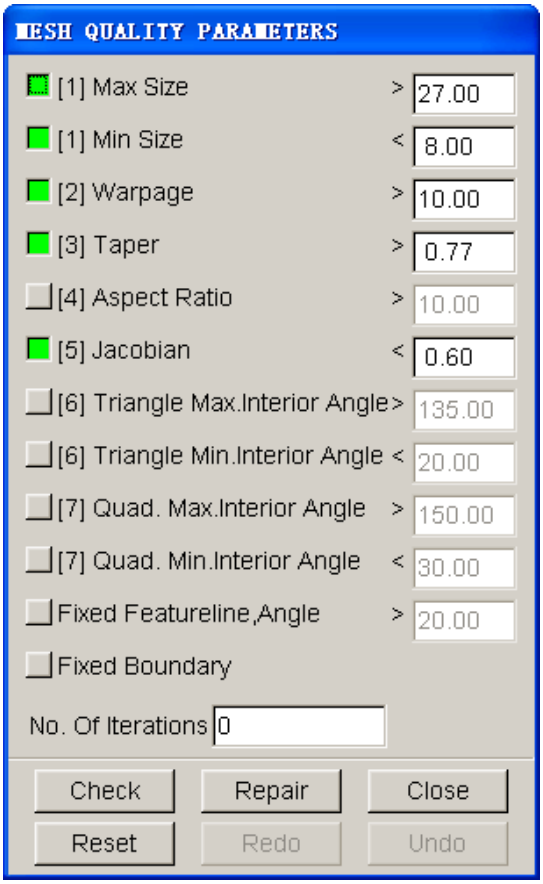
1. VPG prompts:
   > SEARCHING FEATURE LINES…



   - **EXIT** returns the user to the model checker menu.

   Enter **CHECK AND REPAIR**

2. The MESH QUALITY PARAMETERS window appears,



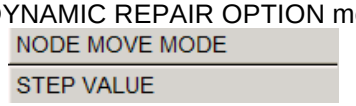
- The user may define parameters. Upon clicking “Check,” VPG prompts:
  > xxxx ELEMENTS FAILED XXXXXX CHECK
  > PICK A NODE WILL INITIATE DYNAMIC REPAIR

Enter **SELECT MESH**

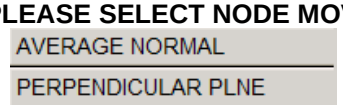
2. VPG prompts:
   > SELECT ELEMENTS
   - After entering DONE, the MESH QUALITY PARAMETERS window pops up.

Enter **SET DYNAMIC REPAIR**

2. The DYNAMIC REPAIR OPTION menu pops up, as following:



3. If entering NODE MOVE MODE, VPG prompts:
   > PLEASE SELECT NODE MOVE MODE



4. If entering STEP VALUE, VPG prompts:
   > PLEASE INPUT STEP VALUE (DEFAULT = 1.0)

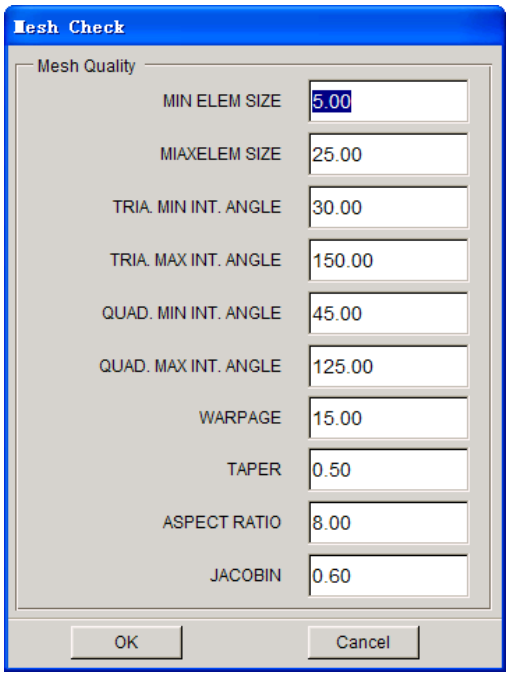
6.5.25 SUMMARY CHECK

Description: The SUMMARY CHECK function performs a comprehensive model check on the current elements in the model database, using user-specified parameters. The results of the SUMMARY CHECK function are displayed as a table of model quality parameters by parts as a percentage of elements passing each modeling criteria value.

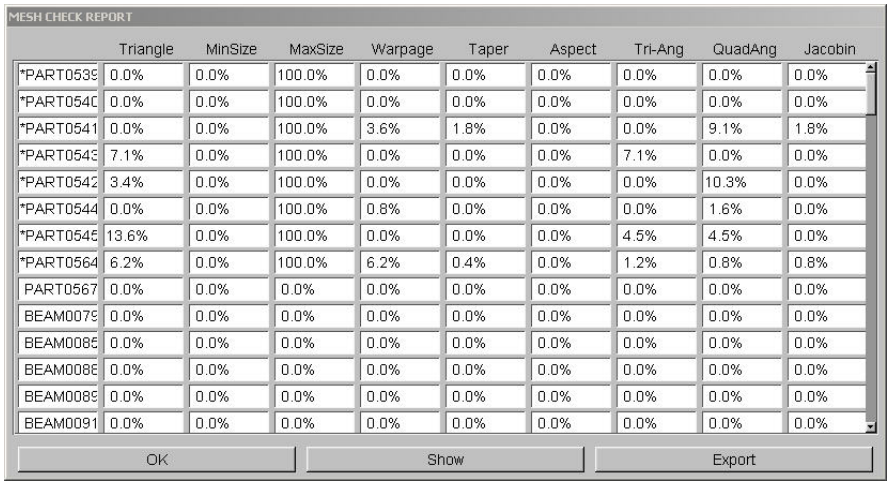
This data may be exported into an ASCII file for model documentation.

Usage:

1. VPG prompts:
   > INPUT MESH CHECK CATEGORY
   - Simultaneously, the “Mesh Check” window is displayed, as follows:



   - The user may modify any of the model checking parameters in the table and selects OK. A MESH CHECK REPORT window is displayed, summarizing the model mesh quality by parts.



6.5.26 GAP REPAIR

Description: This function will fill in the gaps between selected elements.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS

2. After selecting elements and click DONE, VPG prompts:
   > ACCEPT REPAIRING RESUTL?
   ? YES
   NO
   - If **YES** is selected, gap between elements will be repaired, free nodes will be generated after merging elements
   - If **NO** is selected, nothing is done.

6.5.27 TIME STEP

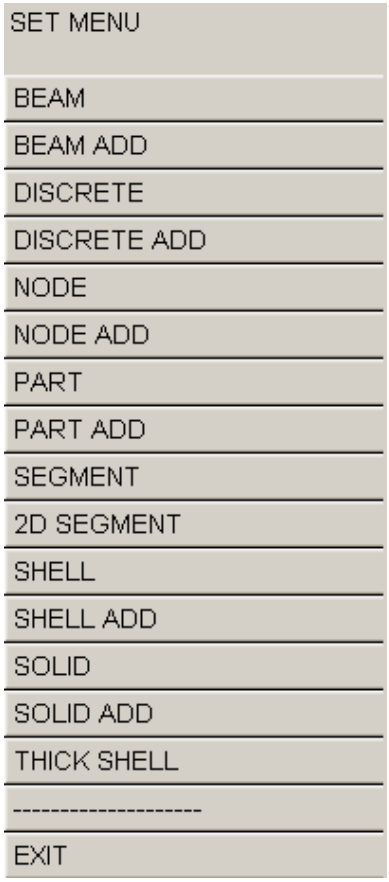
Description: the 10 smallest timestep table will pop up for user to confirm, also the 10 elements have smallest steps will be labeled, which is convenient for user to identify which elements have small time step, and will give user a reference to define timestep.

Note: This timestep calculation is just based on shell elements and MAT1、 MAT3、MAT18、MAT24、MAT26、MAT36、MAT37、MAT39、MAT64、MAT103、MAT113、MAT122、MAT123、MAT125、MAT133 by far, will ignore other types elements.

6.6 SET MENU (LS-DYNA)

The functions in the SET MENU create sets that facilitate the organization of data specification and output control data.

The following options are available in the SET MENU when the analysis code is set to LS-DYNA:



Note: When the analysis code is set to NASTRAN, see section 6.7.

Each option contains a similar submenu with the following options for manipulating the output control of their respective set groups:

ANNOTATE
CREATE
CURRENT
DELETE
REMOVE EMPTY SETS
REMOVE UNREF. SETS
INCLUDE (ELEMENTS/NODES/PARTS...)
LIST
REMOVE (ELEMENTS/NODES/PARTS...)
RENUMBER SET
REVIEW LIST
SHOW SET
SET ON/OFF (PART)
VIEW

EXIT

A detailed description of each function is given in the following sections.

6.6.1 ANNOTATE

Description: This function allows the user to add notes to a selected set.

Usage:

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   ? [List of all sets]
   CURSOR AT (element/node/part)
   EXIT

2. After selecting a set, VPG prompts:
   > SET X IS SELECTED
   > ENTER CARD DESCRIPTION (70 CHAR. MAX.)
   - Once the note is entered, the user is returned to step 1.

6.6.2 CREATE

Description: This function allows the user to generate sets with unique ID numbers. The respective element/node/part may be included in multiple sets.

Usage:

1. VPG prompts:
   > ENTER NEW (type) SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.
   - If the user enters a set ID number already assigned, VPG returns the prompt:
     > (type) SET xxxx ALREADY EXISTS

2. VPG prompts:
   > SELECT (element/node/part)
   - The user may press the left most mouse button or "N" to select the element nearest to the cursor.
   - If the user selects the same element/node/part twice while creating a the set, VPG returns the prompt:
     > (type) xxxx ALREADY PICKED
   - If the user selects DONE/EXIT, VPG saves the set and prompts:
     > X (type) INCLUDED IN (type) SET X
   - If the user selects ABORT, VPG deletes the node set and returns the user to the SET submenu.

6.6.3 CURRENT

Description: This function allows the user to select the current set from a list of previously defined sets.

Usage:

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   ? [List of all sets]

CURSOR AT (element/node/part)
EXIT

2. Once the set(s) is selected, VPG prompts:
   > SET X FLAGGED AS CURRENT SET.
   • **The selected set is current.**

6.6.4 DELETE

Description: This function allows the user to remove sets by ID number. Selecting the set name also indicates where the set has been used such as in boundary conditions. (VPG is specifically configured to recognize sets for this procedure.)

Usage:

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   • **[List of all sets]**
   CURSOR AT (element/node/part)
   ALL (type) SETS
   EXIT

2. If the user selects ALL (type) SETS in step 2, the sets are highlighted in red in the VPG MENU WINDOW.
   > OK TO DELETE ALL (type) SETS? (Y/N)
   ? YES
   NO
   • If YES, VPG issues the following message:
   > ALL (type) SETS DELETED FROM DATABASE.
   • If NO, VPG returns the user to the set submenu.

3. If the user selects an individual set in step 1, the set will be highlighted in red. When the user exits the submenu, all highlighted sets will be deleted.

6.6.5 REMOVE EMPTY SETS (NODE)

Description: This function deletes sets with no nodes assigned to them.

Usage:

1. VPG deletes any empty node sets. If empty node sets are found, VPG shows the number of sets found and prompts:
   >OK TO DELETE (1) EMPTY NODE SETS? (Y/N)
   If no unreferenced node sets are found, VPG prompts:
   >NO EMTPY NODE SETS FOUND
   and returns to the node set menu.

6.6.6 REMOVE UNREF. SETS (NODE)

Description: This function deletes sets that have not been assigned boundary conditions, constraints, initial conditions, loads, or contacts.

Usage:

1. VPG deletes any unreferenced node sets. If unreferenced node sets are found,

VPG shows the number of sets found and prompts:
>OK TO DELETE (1) UNREFERENCED NODE SETS? (Y/N)
If no unreferenced node sets are found, VPG prompts:
>NO UNREF. NODE SETS FOUND
and returns to the node set menu.

6.6.7 INCLUDE

Description: This function adds additional elements/nodes/parts to the current set.

Usage:

1. VPG prompts:
   > SELECT (element/node/part) FOR (type) SET XXXX
   > SELECT (element/node/part)

2. Once the elements/nodes/parts have been selected and the user has exited the INCLUDE submenu, VPG displays the new additions with their set number and prompts:
   > XXXX (ELEMENTS/NODES/PARTS) IN SET XXXX

6.6.8 LIST

Description: This function displays a list of the defined sets in the model database.

6.6.9 REMOVE

Description: This function removes elements/nodes/parts from the current set.

Usage:

1. VPG prompts:
   > SELECT (element/node/part) FROM (type) SET XXXX
   > SELECT (element/node/part)

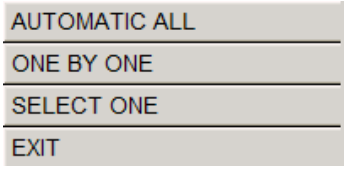
2. Once the elements/nodes/parts have been selected and the user has exited the REMOVE submenu, VPG removes the elements/nodes/part from their set.

6.6.10 RENUMBER SET

Description: This function allows the user to renumber any of the defined sets. This may be performed on a single set or an all sets contained in the database.

Usage:

1. VPG prompts:
   > SELECTED OPTIONS



2. If entering AUTOMATIC ALL, VPG prompts:
   > ENTER FIRST (type) ID & INC., TYPE 0 TO EXIT

3. If entering ONE BY ONE, VPG prompts:
   > ENTER NEW (type) SET ID (OLD= X), TYPE 0 TO EXIT
   • This prompt is repeated until all sets are given a new set ID.

4. If entering SELECT ONE, VPG prompts:
   > SELECT SET FROM (type) SET LIST
   • Once set is selected, VPG prompts:
   > SET X IS SELECTED
   > ENTER NEW (type) SET ID (OLD= X), TYPE 0 TO EXIT

6.6.11 REVIEW LIST

Description: This function displays the defined sets in the MENU WINDOW. When a set is selected, the elements/nodes/parts of that set are displayed in the CONTENTS OF SET window.

6.6.12 SHOW SET

Description: This function allows the user to display all or some of the defined sets. VPG displays an ID number next to each element/node/part.

Usage:

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   ? [List of all sets]
   CURSOR AT (element/node/part)
   ALL (type) SETS
   EXIT
   • VPG displays all node sets and returns the user to the set submenu.

Note: Because more than one set ID number may be assigned to each element/node/part, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific set at a time.

6.6.13 SET ON/OFF (PART)

Description: This function allows the user to turn on and off parts included in a selected part set. It allows for convenient organization of parts in a complicated model and manipulation of large numbers of parts in a sub-assembly level.

6.6.14 SET ATTRIBUTE DATA ON/OFF

Description: This function toggles on/off the attribute data option of set, if this option is on, user can define the DA option when creating set.

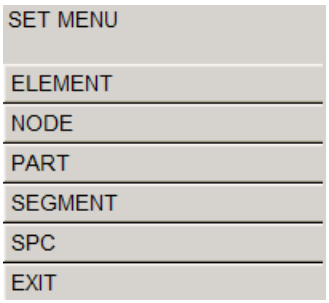
6.6.15 MODIFY SET ATTRIBUTE

Description: This function allows the user to modify DA1-DA4 options in the card of *SET_PART, *SET_NODE, *SET_SHELL, *SET_SEGMENT, and *SET_2D_SEGMENT.

6.7 SET MENU (NASTRAN)

The functions in the SET MENU create sets that facilitate the organization of output control data.

The following options are available in the SET menu when the analysis code is set to NASTRAN in the SETUP MENU:



Note: If the analysis code is set to LS-DYNA, see section 6.6.

Each option contains a similar submenu with the following options for manipulating the output control of their respective set groups:

ANNOTATE
CREATE
CURRENT
DELETE
REMOVE EMPTY SETS
REMOVE UNREF. SETS
INCLUDE (ELEMENTS/NODES/PARTS...)
LIST
REMOVE (ELEMENTS/NODES/PARTS...)
RENUMBER SET
REVIEW LIST
SHOW SET
SET ON/OFF (PART)
VIEW
EXIT

A detailed description of each function is given in the following sections.

6.7.1 ANNOTATE (NODE)

Description: This function allows the user to add notes to a selected set.

Usage:

1. VPG prompts:
   > SELECT SET FROM NODE SET LIST
   ? [List of all sets]
   CURSOR AT NODE
   EXIT

2. After selecting a set, VPG prompts:
   > SET X IS SELECTED
   > ENTER CARD DESCRIPTION (70 CHAR. MAX.)
   - Once the note is entered, the user is returned to step 1.

6.7.2 CREATE (NODE)

Description: This function allows the user to generate sets with unique node and element ID numbers. The nodes may be included in multiple sets.

Usage:

1. VPG prompts:
   > ENTER NEW NODE SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.
   - If the user enters a node set ID number already assigned, VPG returns the prompt:
     > NODE SET xxxx ALREADY EXISTS

2. VPG prompts:
   > SELECT NODES
   - The user may press the left most mouse button or "N" to select the node nearest to the cursor.
   - If the user selects the same node twice while creating a node set, VPG returns the prompt:
     > NODE xxxx ALREADY PICKED
   - If the user enters DONE, VPG saves the set and prompts:
     > X NODES INCLUDED IN NODE SET X
   - If the user selects ABORT, VPG deletes the node set and returns the user to the NODE SET submenu.

6.7.3 CURRENT (NODE)

Description: This function lists the defined node sets in a pop-up window. The user can then select the current set from those listed.

Usage:

1. VPG prompts:
   > SELECT SET FROM NODE SET LIST
   ? [List of all node sets]
   CURSOR AT NODE
   EXIT

2. Once the set(s) is selected, VPG prompts:
   > SET X IS SELECTED
   > SET X FLAGGED AS CURRENT SET.

6.7.4 DELETE (NODE)

Description: This function allows the user to remove node sets by ID number. (VPG is specifically configured to recognize node sets for this procedure. For deleting element sets, refer to the command in section 6.7.16.)

Usage:

1. VPG prompts:
   > SELECT SET FROM NODE SET LIST
   ? [List of all node sets]

CURSOR AT NODE
ALL NODE SETS
UNDO
EXIT

2. If the user selects ALL NODE SETS in step 2, VPG prompts:
   > OK TO DELETE ALL NODE SETS? (Y/N)
   ? YES
   NO
   • If YES, VPG issues the following message:
   > ALL NODE SETS DELETED FROM DATABASE.
   • If NO, VPG returns the user to the NODE SET submenu.

3. If the user selects an individual node set in step 1, VPG prompts:
   > SET X IS SELECTED

4. EXIT returns the user to the NODE SET submenu.

6.7.5 REMOVE EMPTY SETS (NODE)

Description: This function deletes sets with no nodes assigned to them.

6.7.6 REMOVE UNREF. SETS (NODE)

Description: This function deletes sets that have not been assigned boundary conditions, constraints, initial conditions, or loads.

6.7.7 INCLUDE NODES

Description: This function adds additional nodes to the current node set.

Usage:

1. VPG prompts:
   > SELECT NODES FOR NODE SET XXXX
   > SELECT NODES

2. Once the nodes have been selected and the user has exited the INCLUDE submenu, VPG displays the nodes with their node set number

6.7.8 LIST

Description: This function lists the defined sets in a pop-up window.

6.7.9 REMOVE NODES

Description: This function deletes selected nodes from the current node set.

Usage:

1. VPG prompts:
   > SELECT NODES FOR NODE SET XXXX
   > SELECT NODES

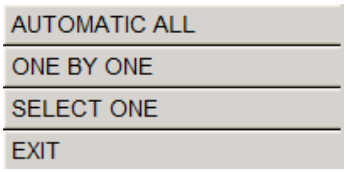
2. Once the nodes have been selected and the user has exited the REMOVE submenu, VPG removes the nodes from their node set.

6.7.10 RENUMBER SET

Description: This function allows the user to renumber any of the defined sets. This may be performed on a single set, or an all sets contained in the database.

Usage:

1. VPG prompts:
   > SELECTED OPTIONS



2. If entering AUTOMATIC ALL, VPG prompts:
   > ENTER FIRST NODE ID & INC., TYPE 0 TO EXIT

3. If entering ONE BY ONE, VPG prompts:
   > ENTER NEW NODE SET ID (OLD= X), TYPE 0 TO EXIT
   • This prompt is repeated until all sets are given a new set ID.

4. If entering SELECT ONE, VPG prompts:
   > SELECT SET FROM NODE SET LIST
   • Once set is selected, VPG prompts:
   > SET X IS SELECTED
   > ENTER NEW NODE SET ID (OLD= X), TYPE 0 TO EXIT

6.7.11 REVIEW LIST

Description: This function displays the defined sets in the MENU WINDOW. When a set is selected, the nodes of that set are displayed in the CONTENTS OF SET window.

6.7.12 SHOW SET (NODE)

Description: This function allows the user to display all or some of the defined node sets. VPG displays an ID number next to each node.

Usage:

1. VPG prompts:
   > SELECT SET FROM NODE SET LIST
   ? [List of all node sets]
   CURSOR AT NODE
   ALL NODE SETS
   UNDO
   EXIT
   • VPG displays all node sets and returns the user to the NODE SET sub-menu.

Note: Because more than one node set ID number may be assigned to each node, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific node set at a time.

6.7.13 ANNOTATE (ELEMENT)

Description: This function allows the user to add notes to a selected set.

Usage:

1. VPG prompts:
   > SELECT SET FROM ELEMENT SET LIST
   ? [List of all sets]
   CURSOR AT ELEMENT
   EXIT

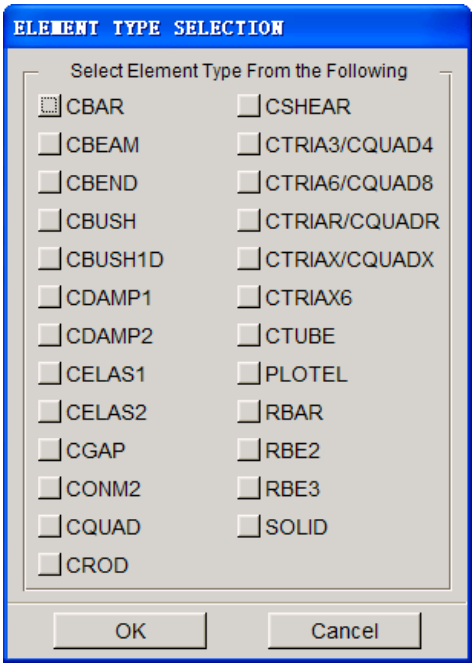
2. After selecting a set, VPG prompts:
   > SET X IS SELECTED
   > ENTER CARD DESCRIPTION (70 CHAR. MAX.)
   - Once the note is entered, the user is returned to step 1.

6.7.14 CREATE (ELEMENT)

Description: This function allows the user to generate sets with unique element ID numbers. The elements may be included in multiple sets.

Usage:

1. The ELEMENT TYPE SELECTION window pops up, as following:

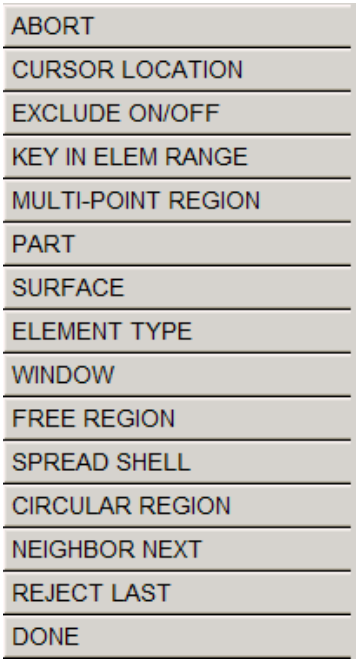


2. After the user selects element set type, VPG prompts:
   > ENTER NEW ELEMENT SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.
   - If the user enters an element set ID number already assigned, VPG returns the prompt:
     > ELEMENT SET xxxx ALREADY EXISTS.

3. After the user enters an element set ID number, VPG prompts:

> **SELECT ELEMENTS**

- Select elements by the following methods:



- The user may press the left mouse button or type "C" to select the element nearest to the cursor.
- If the user selects the same element twice while creating a set, VPG returns the prompt:
  > **ELEMENT xxxx ALREADY PICKED**
- If the user selects DONE, VPG saves the set and prompts:
  > **X ELEMENTS INCLUDED IN ELEMENT SET X**
- If the user selects ABORT, VPG deletes the set and forwards the user to the ELEMENT SET submenu.

Note: Users may not assign the same ID numbers to both node sets and element sets.

6.7.15 CURRENT (ELEMENT)

Description: This function lists the defined element sets in a pop-up window. The user can then select the current set from those listed.

Usage:

1. VPG prompts:
   > **SELECT SET FROM ELEMENT SET LIST**
   ? **[List of all element sets]**
   CURSOR AT ELEMENT
   EXIT

2. Once the set is selected, VPG prompts:
   > **SET X IS SELECTED**
   > **ELEMENT SET X FLAGGED AS CURRENT SET.**

6.7.16 DELETE (ELEMENT)

Description: This function allows the user to remove element sets by ID number. (VPG is specifically configured to recognize element sets for this procedure. For deleting element nodes, refer to the command in section 6.7.4.)

Usage:

1. VPG prompts:
   > SELECT SET FROM ELEMENT SET LIST
   ? [List of all element sets]
   CURSOR AT ELEMENT
   ALL ELEMENT SETS
   UNDO
   EXIT

2. If the user selects ALL ELEMENT SETS in step 1, VPG prompts:
   > OK TO DELETE ALL ELEMENT SETS. (Y/N)
   ? YES
   NO
   - If YES, VPG issues the following message:
     > ALL ELEMENT SETS DELETED FROM DATABASE
   - If NO, VPG returns the user to the ELEMENT SET submenu.

3. If the user selects an individual element set in step 1, VPG prompts:
   > SET X IS SELECTED
   - If the user selects EXIT, VPG deletes the element set.
   - After again selecting EXIT, the user returns to the ELEMENT SET menu.

6.7.17 INCLUDE ELEMENTS

Description: This function adds additional elements to the current element set.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS FOR ELEMENT SET XXXX
   > SELECT ELEMENTS

2. When the user has finished selecting elements and exited the menu, VPG prompts:
   > xxxx ELEMENTS IN SET XXXX
   - The selected elements are added to the set and numbered accordingly.

6.7.18 LIST

Description: This function lists the defined sets in a pop-up window.

6.7.19 REMOVE ELEMENTS

Description: This function deletes elements from the current element set.

Usage:

1. VPG prompts:
   > SELECT ELEMENTS FROM ELEMENT SET XXXX
   > SELECT ELEMENTS

2. When the user has finished selecting elements and has exited the menu, VPG

prompts:

> xxxx ELEMENTS REMOVED FROM SET XXXX

6.7.20 REVIEW LIST

Description: This function displays the defined sets in the MENU WINDOW. When a set is selected, the elements of that set are displayed in the CONTENTS OF SET window.

6.7.21 SHOW SET (ELEMENT)

Description: This function allows the user to display all or some of the defined element sets. VPG displays the appropriate ID numbers on the screen next to each defined element.

Usage:

1. VPG prompts:
   > SELECT SET FROM ELEMENT SET LIST
   ? [List of all element sets]
   CURSOR AT ELEMENT
   ALL ELEMENT SETS
   UNDO
   EXIT
   - VPG displays all element sets and returns the user to the ELEMENT SET submenu.

Note: Because more than one ID number may be assigned to each element, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific element set at a time.

6.7.22 ANNOTATE (PART)

Description: This function allows the user to add notes to a selected set.

Usage:

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all sets]
   CURSOR AT ELEMENT
   EXIT

2. After selecting a set, VPG prompts:
   > SET X IS SELECTED
   > ENTER CARD DESCRIPTION (70 CHAR. MAX.)
   - Once the note is entered, the user is returned to step 1.

6.7.23 CREATE (PART)

Description: This function allows the user to generate part sets with unique element and node ID numbers. The parts may be included in multiple sets.

Usage:

1. VPG prompts:
   > ENTER NEW PART SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.

2. VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART
   [list of all parts]
   - The user may press the left mouse button or type "C" to select the element nearest to the cursor.

3. VPG prompts:
   > PART XXXX IS SELECTED
   > PICK AN ELEMENT OR PART NAME OF A PART
   [list of all parts]

4. When the user finishes and selects EXIT, VPG prompts:
   > X PARTS INCLUDED IN PART SET X
   - VPG returns to the PART SET menu.

6.7.24 CURRENT (PART)

Description: This function lists the defined part sets in a pop-up window. The user can then select the current set from those listed.

Usage:

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all part sets]
   CURSOR AT ELEMENT
   EXIT

2. Once the set is selected, VPG prompts:
   > SET X IS SELECTED
   > CURRENT PART SET IS X

6.7.25 DELETE (PART)

Description: This function allows the user to remove part sets by ID number. (VPG is specifically configured to recognize part sets for this procedure. For deleting part nodes, refer to the command DELETE (NODE) earlier in this section.)

Usage:

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all PART sets]
   CURSOR AT ELEMENT
   ALL PART SETS
   EXIT

2. If the user selects ALL PART SETS in step 1, VPG prompts:
   > OK TO DELETE ALL PART SETS. (Y/N)
   ? YES
   NO
   - If YES, VPG issues the following message:
     > ALL PART SETS DELETED FROM DATABASE
   - If NO, VPG returns the user to the PART SET submenu.

3. If the user selects an individual part set in step 1, VPG prompts:
   > OK TO DELETE PART SET xxxx?
   ? YES
   NO
   - If the user selects YES, VPG deletes the part set and returns the user to step 1.
   - If the user selects NO, VPG retains the part set and returns the user to step 1.

4. EXIT returns the user to the PART SET submenu.

6.7.26 INCLUDE PARTS

Description: This function adds additional parts to the current part set.

Usage:

1. VPG prompts:
   > SELECT PARTS FOR PART SET XXXX
   > PICK AN ELEMENT OR PART NAME OF A PART

2. Once the user has selected a part, VPG prompts:
   > **PART XXXX IS SELECTED**
   - The user is returned to step 1 until EXIT is selected.

6.7.27 LIST

Description: This function lists the defined sets in a pop-up window.

6.7.28 REMOVE PARTS

Description: This function deletes parts from the current part set.

Usage:

1. VPG prompts:
   > SELECT PARTS FROM PART SET XXXX
   > PICK AN ELEMENT OR PART NAME OF A PART

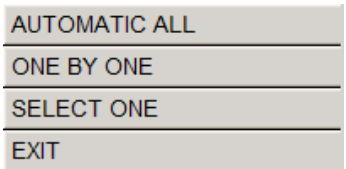
2. When the user has finished selecting parts and has exited the menu, VPG removes the selected parts from the set.

6.7.29 RENUMBER SET

Description: This function renumbers the defined sets.

Usage:

1. VPG prompts:
   > SELECTED OPTIONS



2. If entering AUTOMATIC ALL, VPG prompts:
   > ENTER FIRST PART ID & INC., TYPE 0 TO EXIT

3. If entering ONE BY ONE, VPG prompts:
   > ENTER NEW PART SET ID (OLD= X), TYPE 0 TO EXIT
   • This prompt is repeated until all sets are entered new set ID.

4. If entering SELECT ONE, VPG prompts:
   > SELECT SET FROM PART SET LIST
   • Once set is selected, VPG prompts:
   > SET X IS SELECTED
   > ENTER NEW PART SET ID (OLD= X), TYPE 0 TO EXIT

6.7.30 REVIEW LIST

Description: This function displays the defined sets in the MENU WINDOW. When a set is selected, the parts of that set are displayed in the CONTENTS OF SET window.

6.7.31 SHOW SET (PARTS)

Description: This function allows the user to display all or some of the defined part sets. VPG displays the appropriate ID numbers on the screen next to each defined part.

Usage:

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all part sets]
   CURSOR AT ELEMENT
   EXIT
   • VPG displays the selected part sets and returns the user to the PART SET submenu.

Note: Because more than one ID number may be assigned to each part, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific part set at a time.

6.7.32 SET ON/OFF (PART)

Description: This function allows the user to turn on and off parts included in a selected part set. It allows for convenient organization of parts in a complicated model and manipulation of large numbers of parts in a sub-assembly level.

6.8 BOUNDARY CONDITIONS (LS-DYNA)

The functions of the BOUNDARY CONDITIONS MENU create and verify constraints and loads on finite element models.

A detailed description of each function is given in the following sections.

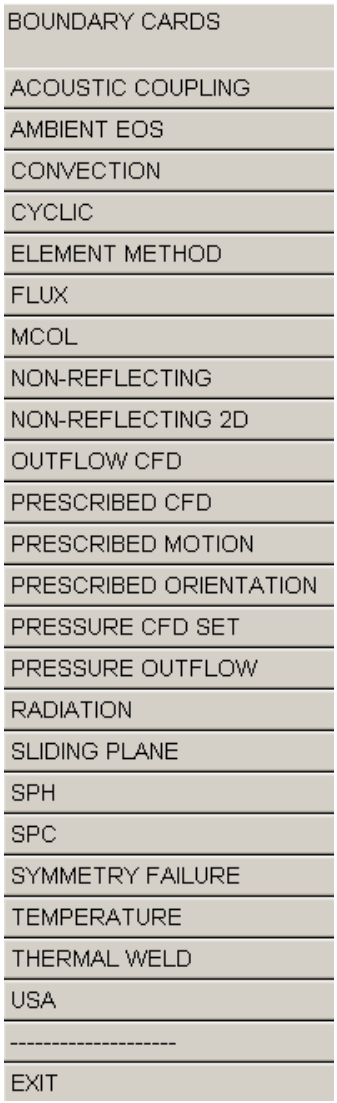
6.8.1 BOUNDARY CARDS

Description: The BOUNDARY CARDS functions create and verify constraints and loads on finite element models. The functions in the BOUNDARY CARDS MENU are organized as follows.

Usage:

1. VPG prompts:

> SELECT OPTION



Note: All of these options share this standard submenu. The submenu yields varying ways to access

and create definition cards for the above BOUNDARY CARDS.

> CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

-BOUNDARY CARDS: ACOUSTIC COUPLING

Description: This function defines acoustic coupling boundary conditions for a thermal or coupled thermal/structural analysis. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: AMBIENT EOS

Description: This function defines ambient Equation of State boundary conditions for a thermal or coupled thermal/structural analysis. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: CONVECTION

Description: This function defines convection boundary conditions for a thermal or coupled thermal/structural analysis. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: CYCLIC

Description: These boundary conditions can be used to model a segment of an object that has rotational symmetry, such as an impeller. The segment boundaries may be curved or planar. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: ELEMENT METHOD

Description: This function defines element method boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: FLUX

Description: This function defines flux boundary conditions for a thermal or coupled thermal/structural analysis. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: MCOL

Description: This function defines MCOL boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: NON-REFLECTING

Description: This function defines a non-reflecting boundary. The option applies to continuum domains modeled with solid elements, as indefinite domains are usually not modeled. For geomechanical problems, this option is important for limiting the size of the models. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: NON-REFLECTING2D

Description: This function defines a non-reflecting boundary. The option applies to continuum domains modeled with two-dimensional solid elements in the XY plane, as indefinite domains are usually not modeled. For geomechanical problems, this option is important for limiting the size of the models. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: OUTFLOW CFD

Description: This function defines outflow CFD boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: PRESCRIBED CFD

Description: This function defines prescribed CFD boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: PRESCRIBED MOTION

Description: This function defines an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also, velocities and displacements can be imposed on rigid bodies. If the local option is active, the motion is prescribed with respect to the local coordinate system for the rigid body. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: PRESCRIBED ORIENTATION

Description: This function defines an imposed orientation for a rigid body. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: PRESSURE CFD SET

Description: This function defines pressure CFD set boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: PRESSURE OUTFLOW

Description: This function defines pressure outflow boundary conditions. These boundary conditions are attached to solid elements using the Eulerian ambient formulation and defined to be pressure outflow ambient elements. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: RADIATION

Description: This function defines radiation boundary conditions for a thermal or coupled thermal/structural analysis. The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: SLIDING PLANE

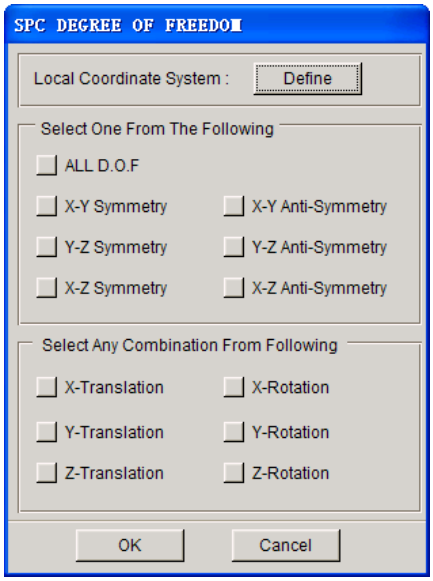
Description: This function defines a sliding symmetry plane. The option applies to continuum domains modeled with solid elements. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: SPH

Description: this function is used to define cards for SPH (Smooth Particle Hydrodynamics)

-BOUNDARY CARDS: SPC

Description: This function defines nodal single point constraints. After selecting CREATE/NODE, VPG displays the following dialogue box. Once the DOF is determined in the dialogue box, VPG prompts the user to select the nodes. DONE finishes the selection and displays the nodes with their SPC set number.



The options in the top and bottom of the SPC DEGREE OF FREEDOM box represent alternate methods for defining the degree of freedom. Only the top or the bottom can be selected, not both.

X-Y SYMMETRY	• Assigns symmetric boundary conditions to the XY plane.
Y-Z SYMMETRY	• Assigns symmetric boundary conditions to the YZ plane.
X-Z SYMMETRY	• Assigns symmetric boundary conditions to the XZ plane.
X-Y ANTI-SYMMETRY	• Assigns asymmetric boundary conditions to the XY plane.
Y-Z ANTI-SYMMETRY	•.Assigns asymmetric boundary conditions to the YZ plane.
X-Z ANTI-SYMMETRY	• Assigns asymmetric boundary conditions to the XZ plane.
X – TRANSLATION	• X-translation will be constrained.
Y – TRANSLATION	• Y-translation will be constrained.
Z – TRANSLATION	• Z-translation will be constrained.
X – ROTATION	• X-rotation will be constrained.
Y – ROTATION	• Y-rotation will be constrained.
Z – ROTATION	• Z-rotation will be constrained.

-BOUNDARY CARDS: SYMMETRY FAILURE

Description: This function defines a symmetry plane with a failure criterion. This option applies to continuum domains modeled with solid elements. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: TEMPERATURE

Description: This function defines temperature boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: THERMAL WELD

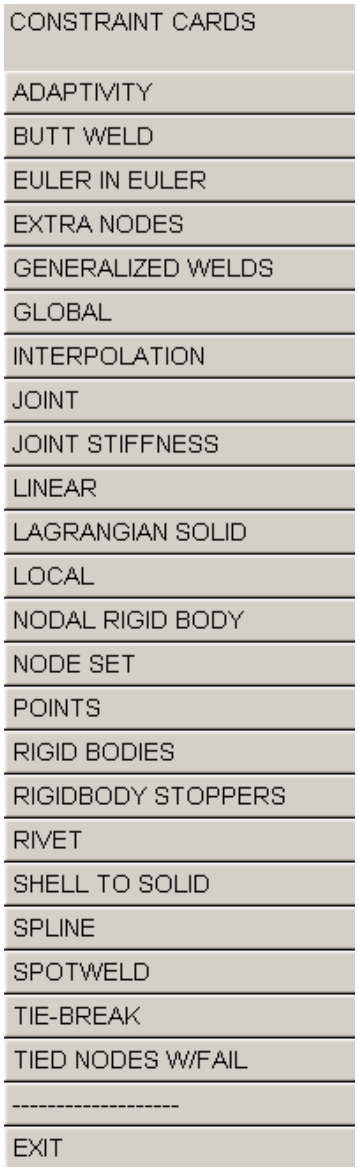
Description: This function defines thermal weld boundary conditions. Refer to the LS-DYNA User's Manual for further description.

-BOUNDARY CARDS: USA

Description: This function defines USA boundary conditions. Refer to the LS-DYNA User's Manual for further description.

6.8.2 CONSTRAINT CARDS

Description: The functions in the CONSTRAINT CARDS menu provide a means of constraining the degrees of freedom in model parts.



Note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE • Defines a new card.
DELETE • Deletes a selected card or cards from those displayed in a pop-up window.
LIST • Lists the cards in the option's database via pop-up window.
MODIFY • Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.
EXIT

-CONSTRAINT CARDS: ADAPTIVITY

Description: This function defines adaptivity.

Usage:

1. When CREATE is selected the CONSTRAINED ADAPTIVITY DEFINITION window pops up.
   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: BUTT WELD

Description: This function defines a butt weld between master node set and slave note set.

Usage:

1. When CREATE is selected the BUTT WELD window pops up.
   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: EULER IN EULER

Description:this fuction is used to define EULER IN EULER card.

Usage:

1. When CREATE is selected the CONSTRAINED EULER IN EULER window pops up.
   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: EXTRA NODES

Description: This function defines extra nodes for a rigid body

Usage:

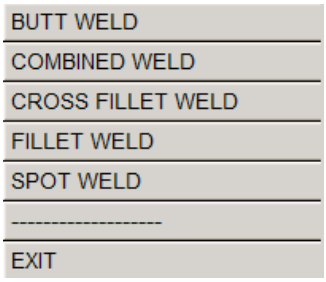
1. When CREATE is selected, VPG prompts:
   > SELECT OPTION
   NODE
   SET
   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: GENERALIZED WELD

Description: This function defines spot and fillet welds. Coincident nodes are permitted if the local coordinate ID is defined. For the SPOTWELD, a local coordinate ID is not required if the nodes are offset. Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criterion is met. The welds may undergo large rotations since the equations of rigid body mechanics are used to update their motion.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: GLOBAL

Description: This function defines global constraints.

Usage:

1. When CREATE is selected, the CONSTRAINED GLOBAL DEFINITION window pops up.
   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: INTERPOLATION

Description: This function defines an interpolation constraint. With this constraint type, the motion of a single dependent node is interpolated from the motion of a set of independent nodes. Please refer to section 6.3.4 for details.

-CONSTRAINT CARDS: JOINT

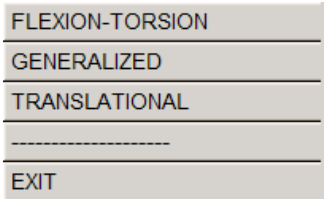
Description: This function defines joint stiffness for joints defined by the previous option.

-CONSTRAINT CARDS: JOINT STIFFNESS

Description: This function defines a joint between two rigid bodies, please refer to section 6.3.4 for details.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: LINEAR

Description: This function defines linear constraint equations between displacements/rotations, which can be defined in local/global coordinate systems.

Usage:

1. When **CREATE** is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: LAGRANGIAN SOLID

Description: This function couples Lagrangian mesh shells, solids or beams to the material points of a Eulerain flow (master). This option may also be used to model rebar in concrete or tire cords in rubber. The slave part or slave part set is coupled to the master part or master part set.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: POINTS

Description: This function defines constrained points.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: LOCAL

Description: This function defines a boundary constraint in a local coordinate system.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: NODAL RIGID BODY

Description: This function defines a rigid body that consists of defined nodes. Refer to section 6.3.4 for more details.

-CONSTRAINT CARDS: NODE SET

Description: This function defines nodal constraint sets for translation motion in global coordinates. Refer to section 6.3.4 for more details.

-CONSTRAINT CARDS: RIGID BODIES

Description: This function merges two rigid bodies. One rigid body, the slave rigid body, is merged to the other one, called the master rigid body.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA

User's Manual for further description.

-CONSTRAINT CARDS: RIGID BODY STOPPERS

Description: This function defines constrained rigid body stoppers.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: RIVET

Description: This function defines constrained rivets.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: SHELL TO SOLID

Description: This function defines a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: SPLINE

Description: This function defines an elastic cubic spline interpolation constraint.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: SPOTWELD

Description: This function defines massless spot welds between non-contiguous nodal pairs. Refer to section 6.3.4 for more details.

-CONSTRAINT CARDS: TIE-BREAK

Description: This function defines a tied shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-CONSTRAINT CARDS: TIED NODES W/FAIL

Description: This function defines a tied node set with the failure based on plastic strain. The nodes must be coincident.

Usage:

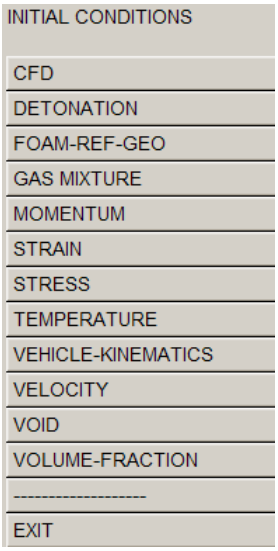
1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

6.8.3 INIT. COND. CARDS

Description: Initial condition cards provide a way of initializing velocities and detonation points.

Usage:

1. VPG prompts:
   > SELECT OPTION



Note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE • Defines a new card.
DELETE • Deletes a selected card or cards from those displayed in a pop-up window.
LIST • Lists the cards in the option's database via pop-up window.
MODIFY • Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.
EXIT

-INIT. COND. CARDS: CFD

Description: This card defines initial CFD. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: DETONATION

Description: This card defines points to initiate the location of high explosive detonations in part IDs that use material type 8. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: FOAM-REF-GEO

Description: This card defines initial foam reference geometry. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: GAS MIXTURE

Description: This function is used to initialize the temperature and the density of different gas species in *MAT_GAS_MIXTURE. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: MOMENTUM

Description: This card defines initial momentum to be deposited in solid elements. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: STRAIN

Description: This card defines initial strain. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: STRESS

Description: This card defines initial stress and plastic strains in the Hughes-Liu beam elements. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: TEMPERATURE

Description: This card defines initial nodal point temperatures using nodal set ID's or node numbers. These initial temperatures are used in a thermal-only analysis or a coupled thermal/ structural analysis. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: VEHICLE-KINEMATICS

Description: This card defines initial vehicle kinematics. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: VELOCITY

Description: This card defines initial nodal point translational velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: VOID

Description: This card defines initial voided part set ID's or part numbers. Refer to the LS-DYNA User's Manual for further description.

-INIT. COND. CARDS: VOLUME-FRACTION

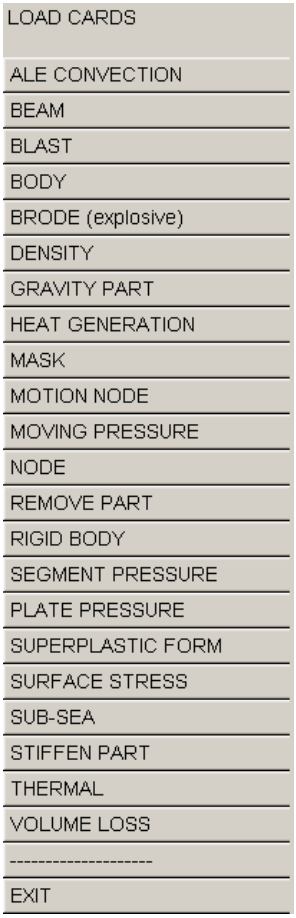
Description: This card defines initial volume fraction. Refer to the LS-DYNA User's Manual for further description.

6.8.4 LOAD CARDS

Description: The LOAD CARD submenu contains cards that define applied forces.

Usage:

1. VPG prompts:
   > SELECT OPTION



Note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE • Defines a new card.
DELETE • Deletes a selected card or cards from those displayed in a pop-up window.
LIST • Lists the cards in the option's database via pop-up window.
MODIFY • Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.
EXIT

-LOAD CARDS: ALE CONVECTION

Description: This card defines the convection thermal energy transfer from a hot ALE fluid to the surrounding Lagrangian structure. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: BEAM

Description: This card defines a distributed traction load along any local axis, beam, or set of beams. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: BLAST

Description: This card defines a blast load. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: BODY

Description: This card defines body force loads due to a prescribed base acceleration or angular velocity using global axes directions. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: BRODE

Description: This card defines pressure loads due to explosion. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: DENSITY

Description: This card defines density versus depth for gravity loading. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: GRAVITY PART

Description: This card defines gravity for individual parts. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: HEAT GENERATION

Description: This card defines solid elements or solid element sets with heat generation. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: MASK

Description: This card defines mask load. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: MOTION NODE

Description: This card defines a concentrated nodal force or moment to a node based on the motion of another node. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: MOVING PRESSURE

Description: This card defines moving pressure loads to a surface. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: NODE

Description: This card defines a concentrated nodal force to a node or a set of nodes. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: REMOVE PART

Description: This card defines to delete the elements of a part. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: RIGID BODY

Description: This card applies a concentrated nodal force to a rigid body. The force is applied at the center of mass, or a moment is applied around a global axis. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: SEGMENT PRESSURE

Description: This card defines a distributed pressure load over one triangular or quadrilateral segment defined by four nodes. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: PLATE PRESSURE

Description: This card defines a distributed pressure load over one shell element or shell element set. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: SUPERPLASTIC FORM

Description: This card is used to perform superplastic forming analyses. This option can be applied to both solid and shell elements. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: SURFACE STRESS

Description: This card is used to Store segment pressures from contact and applied pressure loads on the upper and lower surfaces of the shell surface. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: SUB-SEA

Description: This card defines the effects of the primary explosion and the subsequent bubble oscillations. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: SFIFFEN PART

Description: This card defines for staged construction. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: THERMAL

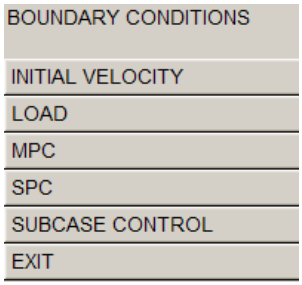
Description: This card defines nodal temperatures that thermally load the structure. Refer to the LS-DYNA User's Manual for further description.

-LOAD CARDS: VOLUME LOSS

Description: This card defines to represent the effect of tunneling on surrounding structures; it is common to assume that a pre-defined fraction of the volume occupied by the tunnel is lost during the construction process. Refer to the LS-DYNA User's Manual for further description.

6.9 BOUNDARY CONDITIONS (NASTRAN)

The functions of the BOUNDARY CONDITIONS MENU create and verify constraints and loads on finite element models. The functions in the BOUNDARY CONDITIONS MENU are organized as follows:



A detailed description of each function is given in the following sections.

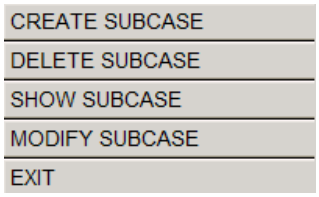
Note: If the selected analysis code is LS-DYNA, a separate menu with different options will appear. See Section 6.8 BOUNDARY CONDITIONS (LS-DYNA).

6.9.1 SUBCASE CONTROL

Description: This function allows users to create and manipulate NASTRAN subcases. Specific descriptions of each of the commands are provided in the following sections.

Usage:

1. VPG prompts:
   > SELECT OPTION



-SUBCASE CONTROL: CREATE SUBCASE

Description: This function incorporates load sets and SPC sets into a subcase. The user may use any combination of load sets and SPC sets in each subcase.

Usage:

1. The NASTRAN SUBCASE DEFINITION window pops up and VPG prompts the user to enter a series of identifiers.
   - The subcase number defaults to the next available set number in sequence.
   - The user enters the name of the subcase.
   - The user can choose from a list of all available SPC sets. If there are no SPC sets defined in the database, then VPG will enter the CREATE SPC SET menu and prompt the user to define the SPC set.

-SUBCASE CONTROL: DELETE SUBCASE

Description: This function deletes selected subcases from the VPG database.

Usage:

1. VPG prompts:
   > SELECT SUBCASE CARD TO DELETE
   • All available subcases are listed in the SUBCASE CARD LIST window.

Note: DELETE SUBCASE does not delete the SPC and Load sets in the subcase.

-SUBCASE CONTROL: SHOW SUBCASE

Description: This function displays the SPC and Load sets in the selected subcases in the VPG database.

Usage:

1. VPG prompts:
   > SELECT SUBCASE CARD TO SHOW
   • All available subcases are listed in the SUBCASE CARD LIST window.

-SUBCASE CONTROL: MODIFY SUBCASE

Description: This function modifies selected subcases in the VPG database.

Usage:

1. VPG prompts:
   > SELECT SUBCASE CARD TO MODIFY
   • All available subcases are listed in the SUBCASE CARD LIST window.

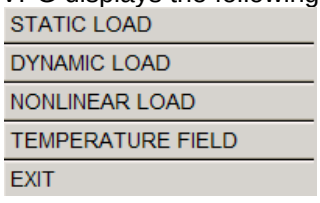
2. Once the user selects a subcase to modify, the NASTRAN SUBCASE DEFINITION window pops up.

6.9.2 LOAD

Description: This menu lists the options to configure and modify load sets in the VPG database. Descriptions of the various commands in this function are provided in the following sections.

Usage:

1. VPG displays the following list of commands in the options area:



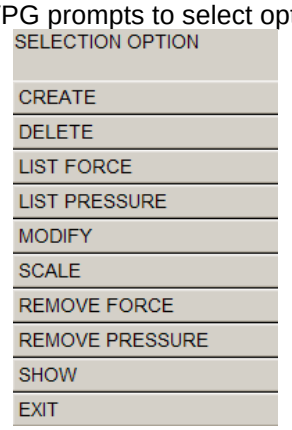
2. These options are described as subsections in the following pages.

-LOAD: STATIC LOAD

Description: This function defines and modifies static loads. The loads may be used in the subcase definition.

Usage:

1. VPG prompts to select option, as following:

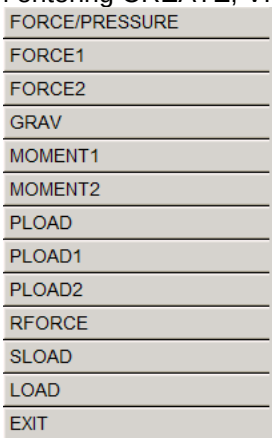


-LOAD: STATIC LOAD: CREATE

Description: This function defines a new load set. The load set may be used in the subcase definition.

Usage:

1. When entering CREATE, VPG prompts to select option, as following:



- The user may enter an integer as the load set number. If a load set with the entered number exists, the user is prompted to enter a different integer.

- The user is allowed to apply pressure loads to the element centers in the VPG model. Distributed element pressure loads are applied normal to the element surface. Positive pressure loads are in the same direction as the positive element normal according to the right hand rule. Negative pressure loads are in the opposite direction of the positive element normal.

-LOAD: STATIC LOAD: DELETE

Description: This function deletes the selected load set.

Usage:

1. VPG prompts:

> SELECT CARD TO DELETE

- Available load sets are displayed in the STATIC LOAD CARD LIST window.

-LOAD: STATIC LOAD: LIST FORCE

Description: This function displays the applied forces and lists node numbers and respective loads in the command area of the screen.

Usage:

1. VPG prompts:
   > SELECT CARD TO LIST

2. VPG prompts:
   > SELECT NODE FOR FORCE
   ? ALL NODES
   - All nodes with applied forces will have arrowheads displayed, but no specific load magnitudes will be listed.
     NODE
     EXIT
   - Pressing the left mouse button or typing N will list the applied force of the grid nearest to the cursor.
   - If no FORCES or MOMENTS are applied at the selected grid, VPG will display the message:
     > NO LOAD SPECIFIED AT SELECTED NODE
     and remain in the LIST FORCE option.
   - EXIT will exit the function LIST FORCE.

-LOAD: STATIC LOAD: LIST PRESSURE

Description: This function displays the pressure loads and respective element numbers in the command area of the screen.

Usage:

1. VPG prompts:
   > SELECT CARD TO LIST

2. VPG prompts:
   > SELECT ELEMENT FOR PRESSURE LOAD
   ? CURSOR LOCATION
   EXIT
   - If no pressure is applied at the selected element, VPG will display the message:
     > NO PRESSURE DEFINED AT SELECTED ELEMENT
     and remain in the LIST PRESSURE option.
   - EXIT will exit the function LIST PRESSURE.
   - If the selected element has a pressure load, VPG prompts:
     > PRESSURE LOAD AT EL. xxxx = xxxx.xx

-LOAD: STATIC LOAD: MODIFY

Description: This function modifies the selected load set.

Usage:

1. VPG prompts:
   > SELECT CARD TO MODIFY
   - Available load sets are displayed in the STATIC LOAD CARD LIST window.

-LOAD: STATIC LOAD: REMOVE FORCE

Description: This function removes nodal forces from selected grids in the current load set.

Usage:

1. VPG prompts:
   > SELECT CARD TO LIST

2. VPG prompts:
   > REMOVE NODAL FORCE
   SELECT NODES
   - The left mouse button may be used to remove selected nodal forces one at a time.
   - When a nodal force is removed, the node is highlighted by a small circle.
   - If a node is selected twice, the following message appears:
     > NODE xxxx ALREADY PICKED
   - DONE removes the selected nodal forces and exits the function REMOVE FORCE.
   - ABORT exits the function REMOVE FORCE without deleting the selected nodal forces.

-LOAD: STATIC LOAD: REMOVE PRESSURE

Description: This function removes pressure loads from selected elements in the current load set.

Usage:

1. VPG prompts:
   > SELECT CARD TO LIST

2. VPG prompts:
   > REMOVE PRESSURE LOAD
   > SELECT ELEMENTS
   - When a pressure load is removed, the element is highlighted and the arrow is removed.
   - If an element is selected more than once, the following message appears:
   > ELEMENT xxxx ALREADY PICKED
   > SELECT ELEMENTS
   - DONE will exit the user to the LOADING OPTIONS submenu.
   - ABORT will exit the user to the LOADING OPTIONS submenu without removing any of the selected pressure loads.

-LOAD: STATIC LOAD: SHOW

Description: This function shows the selected load set.

Usage:

1. VPG prompts:
   > SELECT CARD TO SHOW

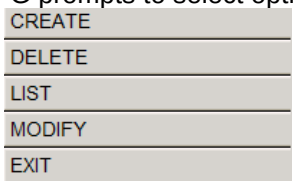
- Available load sets are displayed in the STATIC LOAD CARD window.

-LOAD: DYNAMIC LOAD

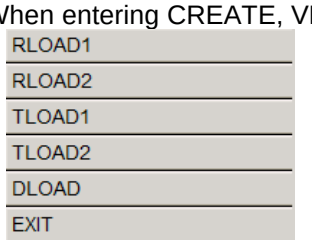
Description: This function defines and modifies dynamic loads. The loads may be used in the subcase definition.

Usage:

1. VPG prompts to select option, as following:



2. When entering CREATE, VPG prompts to select option, as following:



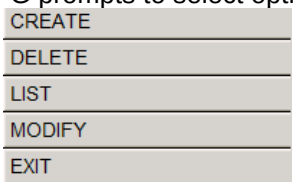
- The functions DELETE, LIST, and MODIFY refer to “STATIC LOAD”.

-LOAD: NONLINEAR LOAD

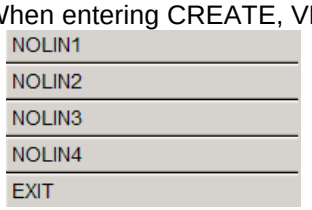
Description: This function defines and modifies nonlinear loads. The loads may be used in the subcase definition.

Usage:

1. VPG prompts to select option, as following:



2. When entering CREATE, VPG prompts to select option, as following:



- The functions DELETE, LIST and MODIFY refer to “STATIC LOAD”.

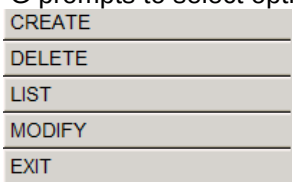
-LOAD: TEMPERATURE FIELD

Description: This function defines and modifies temperature fields. The fields may be used in

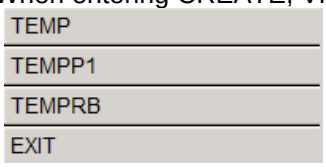
the subcase definition.

Usage:

1. VPG prompts to select option, as following:



2. When entering CREATE, VPG prompts to select option, as following:



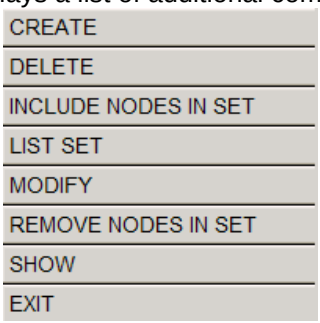
- The functions DELETE, LIST, and MODIFY refer to “STATIC LOAD”.

6.9.3 SPC

Description: This function allows the user to create and/or modify the SPC (single point constraint) options that constrain nodes in six degrees of freedom.

Usage:

1. VPG displays a list of additional commands in the options area:



2. Each of these commands is described in the following pages.

- SPC: CREATE

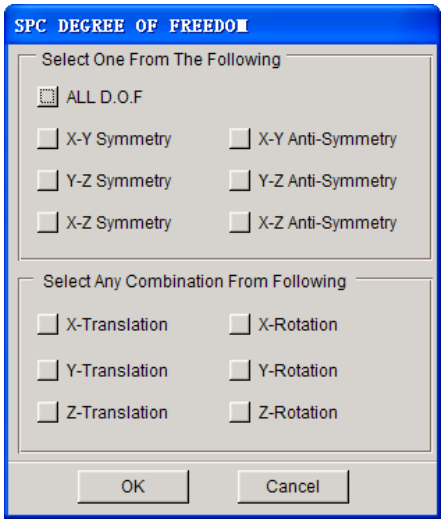
Description: This function creates a new SPC set. The SPC set may be used later for subcase definition.

Usage:

1. When the user enters CREATE/SPC, VPG prompts:
   > ENTER SPC SET NUMBER (= XXXX) OR -1 TO EXIT
   - The user must enter an integer before proceeding.

2. After the user enters SPC set number, VPG prompts:
   > DEFINE D.O.F. FOR SPC
   - Simultaneously, the SPC DEGREE OF FREEDOM box pops up and as

following:



The options in the top and bottom of the SPC DEGREE OF FREEDOM box represent alternate methods for defining the degree of freedom. Only the top or the bottom can be selected, not both.

X-Y SYMMETRY • Assigns symmetric boundary conditions to the XY plane.
Y-Z SYMMETRY • Assigns symmetric boundary conditions to the YZ plane.
X-Z SYMMETRY • Assigns symmetric boundary conditions to the XZ plane.
X-Y ANTI-SYMMETRY • Assigns asymmetric boundary conditions to the XY plane.
Y-Z ANTI-SYMMETRY • Assigns asymmetric boundary conditions to the YZ plane.
X-Z ANTI-SYMMETRY • Assigns asymmetric boundary conditions to the XZ plane.
X – TRANSLATION • X-translation will be constrained.
Y – TRANSLATION • Y-translation will be constrained.
Z – TRANSLATION • Z-translation will be constrained.
X – ROTATION • X-rotation will be constrained.
Y – ROTATION • Y-rotation will be constrained.
Z – ROTATION • Z-rotation will be constrained.

3. Once the DOF is determined in the dialogue box, VPG prompts:
   > D.O.F. xxxx IS FIXED
   > ENTER ENFORCED DISPLACEMENT, DEFAULT IS 0.0

4. VPG prompts:
   > SELECT NODES
   - Nodes may be selected one at a time with the left mouse button.
   - DONE will exit the user to the ENTER SPC SET NUMBER prompt (step 1).
   - ABORT will exit the user to the SPC OPTIONS menu without constraining any of the selected grids.

- SPC: DELETE

Description: This function deletes the selected SPC set.

Usage:

1. VPG prompts:
   > PICK 'SPC' TO DELETE

- All available SPC sets are displayed for selection in the BOUNDARY SPC SET box.

2. When a SPC set has been selected for deletion, VPG prompts:
   > ONE SPC SET IS SELECTED
   - EXIT deletes the SPC set and the user exits the function DELETE.

- SPC: INCLUDE NODES IN SET

Description: This function adds additional nodes into the selected boundary SPC set.

Usage:

1. VPG prompts:
   > PICK A BOUNDARY SPC SET TO ADD NODES

2. VPG prompts:
   > SELECT NODES
   - DONE will add nodes into the SPC set, and the user exits the function INCLUDE NODES IN SET.
   - ABORT will exit the function INCLUDE NODES IN SET without adding any of the selected grids.

- SPC: LIST SET

Description: This function lists the defined sets in a pop-up window.

Usage:

1. VPG prompts:
   > LIST SET ONLY, EXIT TO CONTINUE

- SPC: MODIFY

Description: This function allows the user to simultaneously delete and recreate any boundary SPC set.

Usage:

1. VPG prompts:
   > PICK 'SPC' TO MODIFY

2. After the user selects a boundary SPC set, the BOUNDARY SPC DEFINITION window pops up and the user may modify parameters in the window.

- SPC: REMOVE NODES IN SET

Description: This function deletes nodes from the selected boundary SPC set.

Usage:

1. VPG prompts:
   > PICK A BOUNDARY SPC SET TO REMOVE NODES

2. VPG prompts:

> SELECT NODES

- DONE will release nodes from the SPC set, and the user exits the function REMOVE NODES IN SET.
- ABORT will exit the function REMOVE NODES IN SET without releasing any of the selected grids.

- SPC: SHOW

Description: This function allows the user to display all or some of the defined SPC sets. VPG displays D.O.F. next to each node.

Usage:

1. VPG prompts:
   > PICK 'SPC' TO SHOW
   ? [List of all SPC sets]
   CURSOR AT NODE
   WINDOW
   MULTI-PT REGION
   SELECT ALL
   UNDO ALL
   EXIT

6.9.4 INITIAL VELOCITY

Description: This function allows the user to define and assign both linear and angular velocities to selected nodes.

Usage:

1. VPG displays a list of additional commands in the options area:

2. Each of these commands is described in the following pages.

- INITIAL VELOCITY: CREATE

Description: This function enables the user to define and assign initial velocity to the displayed model.

Usage:

1. After the user enters CREATE, the BOUNDARY BTIC DEFINITION window pops up, and the user sets parameters in the window.

- INITIAL VELOCITY: DELETE

Description: This function enables the user to remove the selected initial velocities from the

database.

Usage:

1. VPG prompts:
   > SELECT CARD TO DELETE
   - All available initial velocities are listed in the TIC CARD LIST box.

- INITIAL VELOCITY: LIST

Description: This function lists the defined initial velocities in a pop-up window.

Usage:

1. VPG prompts:
   > LIST SET ONLY, EXIT TO CONTINUE

- INITIAL VELOCITY: MODIFY

Description: This function allows the user to simultaneously delete and recreate any initial velocity.

Usage:

1. VPG prompts:
   > SELECT CARD TO MODIFY

2. After the user selects an initial velocity, the BOUNDARY BTIC DEFINITION window pops up, and the user can modify parameters in the window.

- INITIAL VELOCITY: SHOW

Description: This function allows the user to display all or some of the defined initial velocities. VPG displays the initial velocities on the screen.

Usage:

1. VPG prompts:
   > SELECT CARD TO SHOW
   ? [List of all initial velocities]
   SELECT ALL
   ABORT
   UNDO
   EXIT

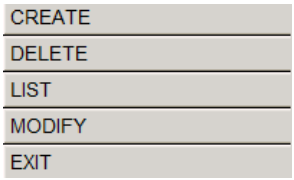
   - All available initial velocities are listed in the TIC CARD LIST box.

6.9.5 MPC

Description: This function allows the user to create and/or modify the MPC.

Usage:

1. VPG displays a list of additional commands in the options area:



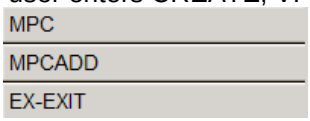
2. Each of these commands is described in the following pages.

- MPC: CREATE

Description: This function allows the user to create the MPC.

Usage:

1. When the user enters CREATE, VPG prompts to select option, as following:



• After the user enters MPC or MPCADD, the BOUNDARY MPC DEFINITION window pops up, and the user can set parameters in the window.

- MPC: DELETE

Description: This function enables the user to remove the selected MPC cards from the database.

Usage:

1. VPG prompts:
   > SELECT CARD TO DELETE
   • All available MPC cards are listed in the MPC CARD LIST box.

- MPC: LIST

Description: This function lists the defined MPC cards in a pop-up window.

Usage:

1. VPG prompts:
   > LIST SET ONLY, EXIT TO CONTINUE

- MPC: MODIFY

Description: This function allows the user to simultaneously delete and recreate any MPC card.

Usage:

1. VPG prompts:
   > SELECT CARD TO MODIFY

2. After the user selects an MPC card, the BOUNDARY MPC DEFINITION window pops up, and the user can modify parameters in the window.

6.10 MATERIAL PROPERTY (LS-DYNA)

The functions in the MATERIAL PROPERTY MENU are designed to define and modify the material properties in the database. Parts are displayed according to the color of their material definition. Parts without material definition are displayed in white.

The functions in this menu are organized as follows:

By switching solver code settings in the ANALYSIS PROGRAM command in the SETUP menu, the user can assign and create materials that are solver-code specific. The user may assign and create materials at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

Note: If the analysis code is set to NASTRAN, see section 6.11.

6.10.1 ADD EROSION

Description: This function creates and adds the erosion to selected materials and modifies the erosion.

Usage:

1. VPG prompts:
   >SELECT OPTION
   CREATE
   - Defines a new card, VPG prompts:
   > SELECT MATERIAL

- After this menu is exited, the Erosion Definition card is shown. The user enters the appropriate figures and selects OK to exit.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

6.10.2 ADD THERMAL_EXPANSION

Description: This function is used to occupy an arbitrary material model in LS-DYNA with a thermal expansion property.

Usage:

1. VPG prompts:

>SELECT OPTION

CREATE

- Definition card is shown. The user enters the appropriate figures and selects OK to exit.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

6.10.3 NONLOCAL

Description: This function is used to define a NONLOCAL failure mode.

Usage:

1. VPG prompts:

>SELECT OPTION

CREATE

- Definition card is shown. The user enters the appropriate figures and selects OK to exit.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

6.10.4 ASSIGN MATERIAL

Description: This function assigns materials to selected parts.

Usage:

1. If no materials are present in the database, VPG gives the message, "NO MATERIALS DEFINED IN DATABASE" and returns to the command prompt.

2. Otherwise, VPG prompts the user to select a material:
   > SELECT A MATERIAL
   • Users may select an existing material by entering one name or by picking an element on the screen that is assigned to the desired material (default).

3. After the user selects the material, VPG prompts:
   > MATERIAL ID (X) IS SELECTED
   > SELECT PARTS TO ASSIGN MATERIAL
   > PICK AN ELEMENT OR PART NAME OF A PART

4. The material will be assigned to the selected elements of the parts. The selected parts will be shown in the material color after the user exits this command.

6.10.5 CHANGE COLOR

Description: This function changes the color of a selected material.

Usage:

1. The color bar is displayed on the right edge of the graphic area.

2. After the user selects the material, VPG prompts:
   > MATERIAL ID (X) IS SELECTED
   > SELECT COLOR FROM COLOR BAR OR EXIT

3. The color of the material changes to the user-selected color from the color bar.

6.10.6 COPY

Description: This function duplicates the properties from one material onto another material.

Usage:

1. VPG displays the material card list and prompts:
   > SELECT MATERIAL TO COPY

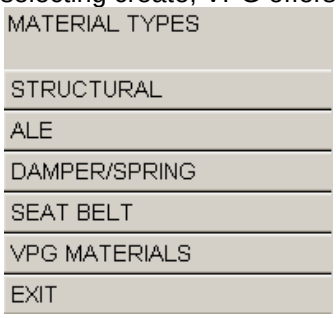
2. Once the user selects a material to copy, VPG copies the material and prompts:
   > MATERIAL ID [xx] DUPLICATED FROM MATERIAL ID [x]

6.10.7 CREATE

Description: This function creates a new material.

Usage:

1. After selecting create, VPG offers the following material categories:



2. Selecting either category yields an LS-DYNA material card list. When the desired material is selected from the list, VPG displays its definition card(s). The user can then edit the material card for the desired properties.

6.10.8 DELETE

Description: This function deletes the user-selected materials from the database. VPG displays the material card list. Selected materials are marked with an asterisk. When the user exits the material card list, the selected materials are deleted.

6.10.9 DEFINE PROPERTIES

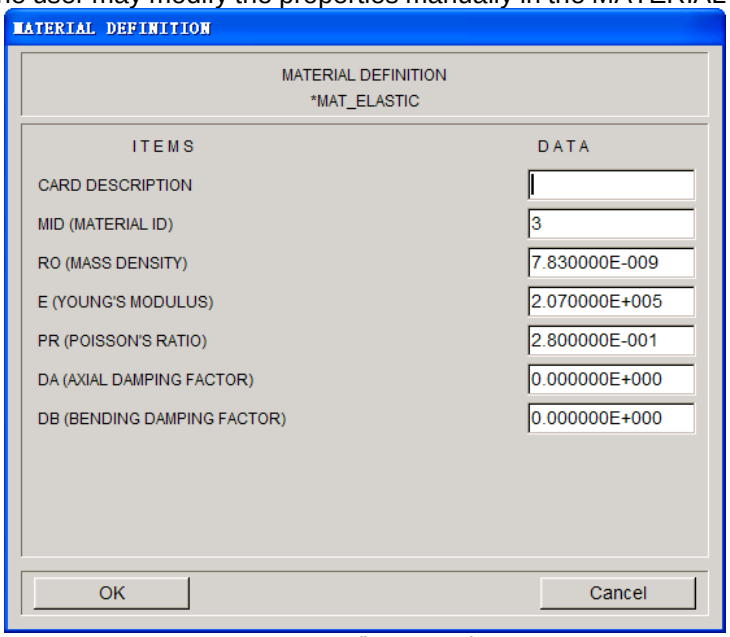
Description: This function defines or modifies the properties of a selected material.

Usage:

1. VPG prompts the user to select a material.
   - Users may select an existing material by picking an element or material name of a material.

2. After the user selects a material, its MATERIAL DEFINITION window appears.

3. The user may modify the properties manually in the MATERIAL DEFINITION window.



Define Properties

6.10.10 ERASE UNREF. MATL

Description: This function automatically deletes all materials that have no parts assigned to them.

Usage:

1. VPG deletes any unreferenced materials. If unreferenced materials are found, VPG shows the number of materials found and prompts:
   >OK TO DELETE (1) UNREFERENCED MATERIALS? (Y/N)
   If no unreferenced materials are found, VPG prompts:
   >NO UNREF. MATERIALS FOUND
   and returns to the material menu.

6.10.11 LIST

Description: This function lists the defined materials in the MATERIAL CARD LIST.

6.10.12 MATERIAL COLOR ON/OFF

Description: This function toggles the display of parts between the material and part color.

6.10.13 ON/OFF

Description: This function toggles the selected material ON/OFF.

Usage:

1. VPG prompts the user to select a material.
   - Users may select an existing material by picking an element or material name of a material.

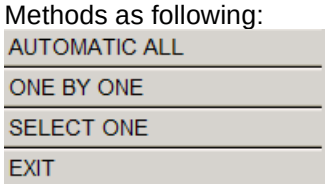
2. The selected material and parts are turned ON/OFF.

6.10.14 RENUMBER

Description: This function allows the user to assign a new material ID (MID) number to a material type.

Usage:

1. VPG prompts:
   > SELECT OPTIONS
   - Methods as following:



6.10.15 SHOW BUSHING

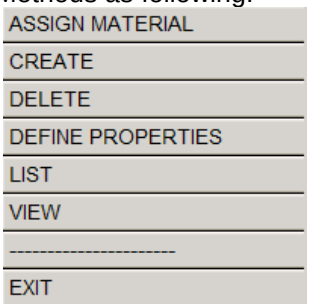
Description: This function shows bushing. Clicking the “Clear screen” icon turns bushing off.

6.10.16 THERMAL MATL MENU

Description: This function creates thermal materials.

Usage:

1. VPG prompts:
   > SELECT OPTION
   - Methods as following:



   - The functions of the THERMAL MATERIAL MENU are similar to the functions of the MATERIAL PROPERTY MENU. See the preceding pages for a complete description of these functions.

6.11 MATERIAL PROPERTY (NASTRAN)

The functions in the MATERIAL PROPERTY MENU are designed to define and modify the material properties in the database. Parts are displayed according to the color of their material definition. Parts without material definition are displayed in white.

The functions in this menu are organized as follows:

By switching solver code settings in the ANALYSIS PROGRAM command in the SETUP menu, the user can assign and create materials that are solver-code specific. The user may assign and create materials at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

Note: If the analysis code is set to LS-DYNA, see section 6.10.

6.11.1 ASSIGN MATERIAL

Description: This function assigns materials to selected parts.

Usage:

1. If no materials are present in the database, VPG gives the message, "NO MATERIALS DEFINED IN DATABASE" and returns to the command prompt.

2. Otherwise, VPG prompts the user to select a material:
   > SELECT A MATERIAL
   - Users may select an existing material by entering one by name or by picking an element on the screen that is assigned to the desired material (default).

3. After the user selects the material, VPG prompts:
   > MATERIAL ID (X) IS SELECTED
   > SELECT PARTS TO ASSIGN MATERIAL
   > PICK AN ELEMENT OR PART NAME OF A PART

4. The material will be assigned to the selected elements of the parts. The selected parts will be shown in the material color after the user exits this command.

6.11.2 CHANGE COLOR

Description: This function changes the color of a selected material.

Usage:

1. The color bar is displayed on the right edge of the graphic area.

2. After the user selects the material, VPG prompts:
   > MATERIAL ID (X) IS SELECTED
   > SELECT COLOR FROM COLOR BAR OR EXIT

3. The color of the material changes to the user-selected color from the color bar.

6.11.3 COPY

Description: This function duplicates the properties from one material onto another material.

Usage:

1. VPG displays the material card list and prompts:
   > SELECT MATERIAL TO COPY

2. Once the user selects a material to copy, VPG copies the material and prompts:
   > MATERIAL ID (X) IS SELECTED
   > MATERIAL ID XX DUPLICATED FROM MATERIAL ID X
   • The user exits the function COPY.

6.11.4 CREATE

Description: This function creates a new material.

Usage:

1. A pop-up window appears with the material types. The list of the material types depends on the analysis program (e.g., NASTRAN, LS-DYNA) defined in the SETUP menu.

2. Once the user has chosen a material type, a pop-up window appears with the material's properties.
   • The displayed property table allows the user to insert values for the material's properties

6.11.5 DELETE

Description: This function deletes the user-selected materials from the database.

Usage:

1. VPG prompts the user for the name of the target material.
   > SELECT MATERIAL
   > PICK AN ELEM (C), OR MATERIAL NAME (M) OF A MATL

2. The user selects the material to be deleted. The elements that are assigned with that material turn gray, and VPG prompts:

> **MATERIAL ID X IS SELECTED**

- If the user enters EXIT, VPG deletes the material and changes the color of the part from gray to white. The white indicates that the part does not have any material properties assigned to it. VPG returns to step 1.

6.11.6 DEFINE PROPERTIES

Description: This function defines or modifies the properties of a selected material.

Usage:

1. VPG prompts the user to select a material.

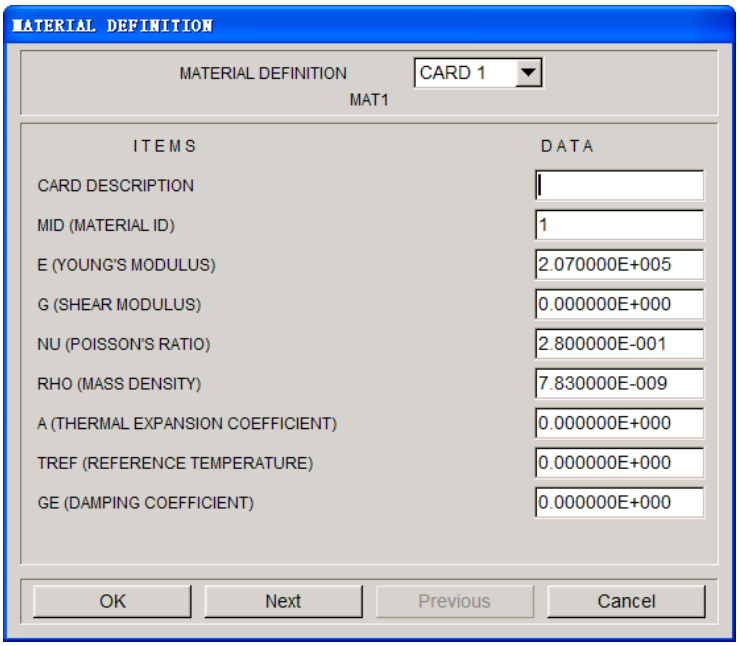
> **SELECT A MATERIAL**

- The user selects a material by picking an element (C) or material name (M) of a material.

2. After the user selects a material, the MATERIAL DEFINITION window appears and a prompt reads:

> **MATERIAL ID (X) IS SELECTED**

3. VPG displays the properties and the type of material that the user has defined in a table. The user may modify the properties in the window.



Define Properties

6.11.7 ERASE UNREF. MATL

Description: This function automatically deletes all materials that have no parts assigned to them.

Usage:

1. VPG deletes any unreferenced materials. If unreferenced materials are found, VPG shows the number of materials found and prompts:
   >OK TO DELETE (1) UNREFERENCED MATERIALS? (Y/N)
   If no unreferenced materials are found, VPG prompts:
   >NO UNREF. MATERIALS FOUND
   and returns to the material menu.

6.11.8 LIST MATERIALS

Description: This function lists the defined materials in the MATERIAL LIST

6.11.9 MATERIAL COLOR ON/OFF

Description: This function toggles the display of parts between the material and part color.

6.11.10 ON/OFF

Description: This function toggles the selected material on or off.

Usage:

1. VPG prompts the user to select material.
   > SELECT MATERIAL
   - The user selects a material by picking an element (C) or material name (M) of a material.

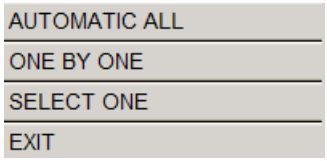
2. The selected material and parts are turned on/off.

6.11.11 RENUMBER ID

Description: This function renumbers the material ID for the selected material or all materials.

Usage:

1. VPG prompts for the option:
   > SELECT OPTIONS



2. If the user enters **AUTOMATIC ALL**, VPG prompts:
   > ENTER FIRST MATERIAL ID. & INC., TYPE 0 TO EXIT

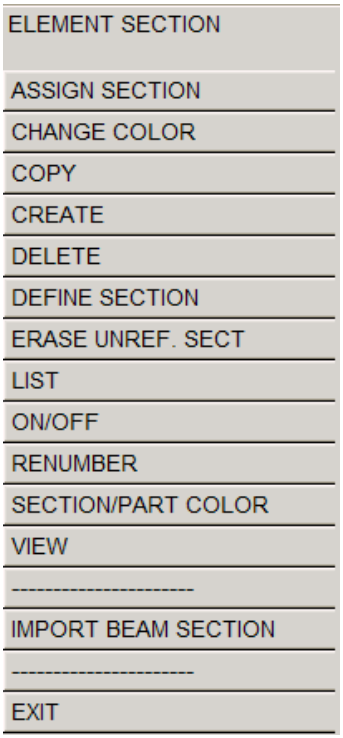
3. If the user selects **ONE BY ONE**, VPG prompts for a new ID for each material. The following prompt will be repeated for each material in the database.
   > ENTER MID [number] FOR MAT [name] OR E TO EXIT
   - If the user enters nothing, VPG defaults to the original values.

4. If the user enters **SELECT ONE**, VPG prompts:
   > PICK AN ELEM(C) OR MATERIAL NAME (M) OF A MATERIAL
   - After the user selects the material, VPG prompts for a new ID.
     > MATERIAL [name] IS SELECTED
     > ENTER MID [= number] FOR MATERIAL [name]

6.12 ELEMENT PROPERTY

The functions in the ELEMENT PROPERTY menu are designed to define and modify the element properties (physical) in the database. When using these commands, parts with property definition are displayed according to their property colors. Parts without property definition are shown in white.

The functions in the ELEMENT PROPERTY menu are organized as follows:



By toggling the solver code settings in the ANALYSIS PROGRAM command in the SETUP window, the user can assign and create properties that are solver-code specific. However, the element property menu and its functions will change slightly between NASTRAN and LS-DYNA analysis. For example, NASTRAN refers to element properties where LS-DYNA refers to element sections.

The user may assign and create properties at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

6.12.1 ASSIGN PROPERTY

Description: This function assigns element properties (physical) to selected parts.

Usage:

1. VPG prompts the user to select a section:
   > SELECT A SECTION
   - The user may select a property by picking an element on the screen that contains a desired property (default) or by selecting a property name
   - If no properties are present in the database, the user will be prompted with the

message, "NO PROPERTIES DEFINED IN DATABASE" and the command prompt will return.

2. VPG prompts the user to designate a part(s) to accept the selected property.
   > SECTION ID (X) IS SELECTED
   > SELECT PARTS TO ASSIGN SECTION
   > PICK AN ELEMENT OR PART NAME OF A PART

3. The property will be assigned to the elements in the selected parts. After the user exits, the selected parts will be shown in the property color.

6.12.2 CHANGE COLOR

Description: This function changes the color of a selected property.

Usage:

1. The color bar is displayed at the right edge of the graphic area.

2. VPG prompts the user to select a section.
   > SELECT SECTION TO CHANGE COLOR
   • The user may select a property by picking an element on the screen that contains a desired property (default) or by selecting a property name

3. After the user selects a section, the prompt reads:
   > SECTION ID (X) IS SELECTED
   > SELECT COLOR FROM COLOR BAR OR EXIT

4. The new color is assigned to the selected section.

6.12.3 COPY (NASTRAN)

Description: This function copies the properties from one element to another.

Usage:

1. VPG prompts the user to select a property.
   > SELECT SECTION TO COPY
   • The user may select a property by picking an element on the screen that contains a desired property (default) or by selecting a property name

2. After the user selects the desired property to copy, VPG prompts:
   > SECTION ID (X) IS SELECTED
   > SECTION ID (XX) DUPLICATED FROM SECTION ID (X)
   • The user exits the function COPY and returns to the ELEMENT PROPERTY menu.

6.12.4 COPY (LS-DYNA)

Description: This function copies the properties from one element to another.

Usage:

1. VPG displays the SECTION CARD LIST and prompts the user to select a section:
   > SELECT SECTION TO COPY

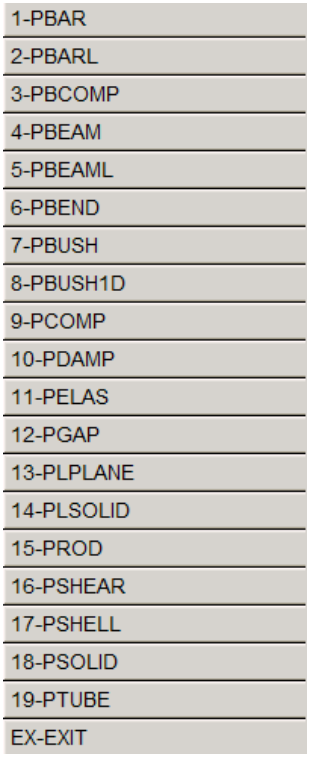
2. After the user selects the desired section to copy, VPG prompts:
   > SECTION ID (XX) DUPLICATED FROM SECTION ID (X)

6.12.5 CREATE (NASTRAN)

Description: This function allows the user to create/define a property.

Usage:

1. VPG prompts the user to select property type.
   > SELECT OPTION
   • The SELECT PROPERTY TYPE menu is shown as below:

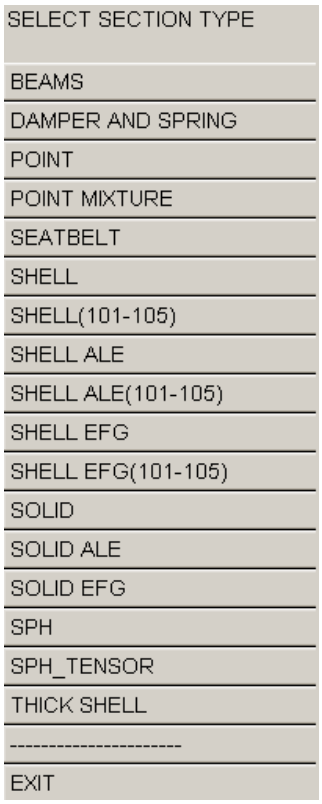


2. Once a property type is selected, VPG displays a definition card in which the user may insert the values for the element properties.

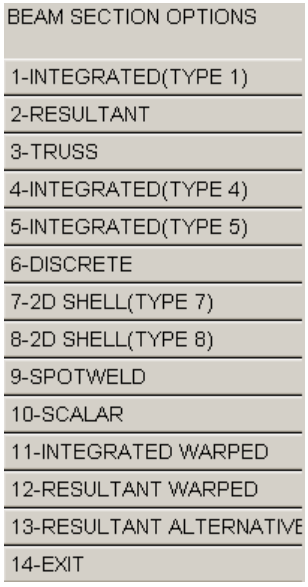
6.12.6 CREATE (LS-DYNA)

Description: This function allows the user to create/define a property. To create a property, VPG first displays a list of section types. Once a type is selected, its definition card is displayed and the user can edit the card for the desired properties.

• The SELECT SECTION TYPE menu is as following:



- If the user enters BEAMS, the BEAM SECTION OPTIONS a menu pops up, as follows:



6.12.7 DELETE (NASTRAN)

Description: This function deletes selected element properties from the database.

Usage:

1. VPG prompts the user for the target property name.

> SELECT SECTION
[LIST OF PROPERTIES]
? WINDOW
MULTI-PT REGION
SELECT ALL
UNDO ALL
UNDO
EXIT

2. After the part is selected, VPG displays the part in gray. The following message is issued:

> SECTION ID X IS SELECTED

- If the user enters EXIT, VPG deletes the property and changes the color of the part from gray to white. White indicates that the part does not have properties assigned to it. VPG returns to step 1.

6.12.8 DELETE (LS-DYNA)

Description: This function deletes selected element sections from the database. Sections are displayed in the section card list. Selecting a section places an asterisk by its name. When the user exits the menu, all asterisked sections will be deleted. The following menu options allow the user to select the desired sections:

- **SECTION NAME** select section by name; also displays section information in dialogue window
- **CURSOR AT ELEMENT** select section by element - default
- **WINDOW** select part by drag window
- **MULTI PT REGION** select part by user defined multi point region
- **SELECT ALL**
- **UNDO ALL**
- **UNDO LAST**

6.12.9 DEFINE PROPERTIES (NASTRAN)

Description: This function allows the user to define or modify the selected property.

Usage:

1. VPG prompts the user to select a property.
   - Select a property by picking an element or property name of a property.

2. After the user selects a property, VPG displays a definition card for editing the element property and prompts:

> SECTION ID (X) IS SELECTED

3. VPG displays the user-defined properties in a table. The user may manually modify the properties and/or their values in the definition card.

6.12.10 DEFINE SECTION (LS-DYNA)

Description: This function allows the user to define or modify the selected property.

Usage:

1. VPG displays the section card list with all defined properties.

2. Once the user selects a section to be redefined, VPG displays its definition card. The user can then redefine that section.

6.12.11 ERASE UNREF. SECT

Description: This function automatically deletes all sections that have no parts assigned to them.

6.12.12 LIST (NASTRAN)

Description: This function lists the names of the element properties in the VPG MENU WINDOW. VPG displays the element properties in their corresponding colors when the properties are turned ON and in white when the properties are turned OFF.

6.12.13 LIST (LS-DYNA)

Description: This function displays the VPG section card list.

6.12.14 ON/OFF

Description: This function turns the selected property and its parts ON/OFF.

Usage:

1. VPG prompts the user to select a property.
   > SELECT SECTION

   WINDOW
   MULTI-PT REGION
   ON ALL SECTION
   OFF ALL SECTION
   UNDO
   EXIT

2. VPG turns the selected property and its parts on/off.

6.12.15 PROPERTY/PART COLOR (toggle)

Description: This function toggles the display of the part between its property and part color.

6.12.16 RENUMBER

Description: This function renumbers the ID for a selected property or for all properties.

Usage:

1. VPG prompts for the option:
   > SELECT OPTIONS

   AUTOMATIC ALL
   ONE BY ONE
   SELECT ONE
   EXIT

2. If the user enters AUTOMATIC ALL, VPG prompts:

> ENTER FIRST PROPERTY ID. & INC., TYPE 0 TO EXIT

3. If the user selects ONE BY ONE, VPG prompts for a new ID for each property. The following prompt will be repeated for each property in the database.
   > ENTER PRID [number] FOR PROPERTY [name] OR E TO EXIT
   • If the user enters nothing, VPG defaults to the original values.

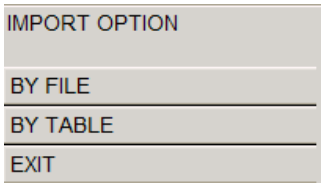
4. If the user enters SELECT ONE, VPG prompts:
   > SELECT PROPERTY NAME
   > ENTER PART NAME OR EXIT
   > PICK AN ELEM(C) OR PROPERTY NAME (P) OF A PROPERTY
   • After the user selects the property, VPG prompts for a new ID.
   > PROPERTY [name] IS SELECTED
   > ENTER PRID [= number] FOR PROP [name]

6.12.17 IMPORT BEAM SECTION

Description: This function defines a beam section by importing from file or table

Usage:

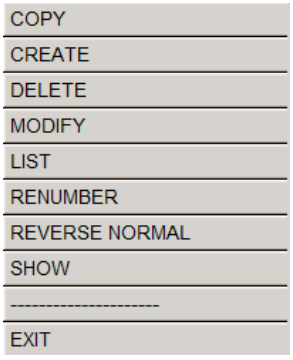
1. VPG prompts for the option:
   > SELECT OPTIONS



- **BY FILE**, VPG opens a file and imports a beam section.
- **BY TABLE**, VPG pops a table, user defines the table for beam section.
- **EXIT**, exits this function.

6.13 CONTACT INTERFACE

The functions in the CONTACT INTERFACE MENU define and modify contact interface data (sliding and stonewall) for transient nonlinear analysis codes. The functions are organized as follows:



Once the CONTACT INTERFACE MENU is selected, the active (ON) interfaces are displayed with the active parts. The master segments are drawn as interior outlines of the elements where the segments are defined. The letter "M" is labeled in the middle of the master segments. Likewise, the slave segments are labeled with the letter "S." The stonewalls are drawn as rectangular shapes, and the slave nodes are labeled with small color-filled squares. The segments, slave nodes, and stonewalls are plotted in the interface color. The segment letters for master and slave may be turned OFF by setting the control button SEGMENT SYMBOL in the SETUP dialog box.

A detailed description of each function is given in the following sections.

Note: Contact interface definition is implemented for a LS-DYNA, RADIOSS, or PAM CRASH analysis only.

6.13.1 COPY

Description: **This function allows the user to copy previously defined interfaces.**

Usage:

1. VPG displays the CONTACT CARD LIST, which lists the defined interfaces and prompts:
   > SELECT CARD TO COPY

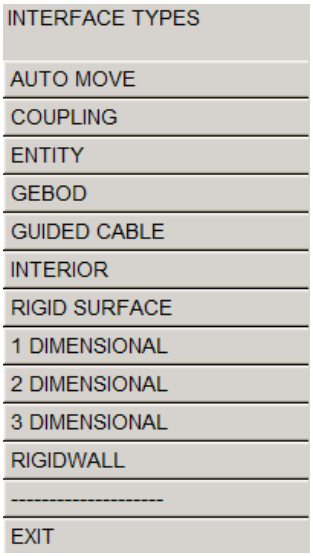
2. Upon selecting an interface card from the menu, VPG prompts:
   > CONTACT ID XX DUPLICATED FROM CONTACT ID X

6.13.2 CREATE

Description: This function allows the user to define new interfaces.

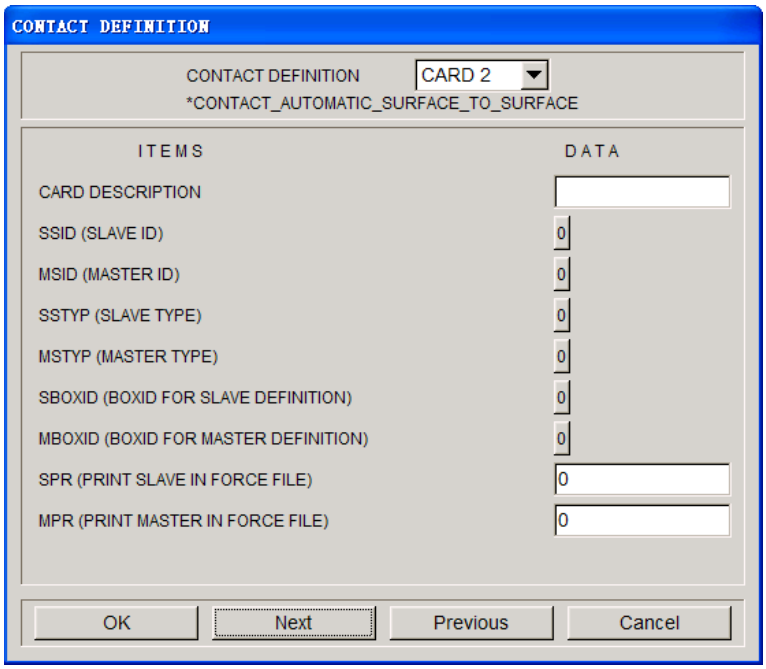
Usage:

1. VPG prompts:
   > SELECT CONTACT OPTION



- VPG supports all contact types in LS-DYNA. If general 3D contact type is selected, the program will display all six cards required to define contact.

  The following notes apply to card 2.



- Unnecessary data fields are in gray and unselectable.
- Default Slave and Master segment type is Segment Set (0). The user needs to define the segment type before defining the Slave and Master Set ID.
- If the Segment Type is modified, the corresponding Segment Set ID needs to be redefined.

6.13.3 DELETE

Description: This function deletes previously defined interfaces.

Usage:

1. VPG prompts:
   > SELECT CARD TO DELETE
   - VPG displays the defined interfaces in the CONTACT CARD LIST with the options:
     > SELECT ALL
     ABORT
     UNDO
     EXIT
   - Cursor pick marks the selected interfaces with an asterisk. Exiting the CONTACT CARD LIST will delete all asterisked interfaces.

6.13.4 MODIFY

Description: This function allows the user to redefine interface properties. Upon selection of the MODIFY command, the CONTACT CARD LIST is displayed with a list of the interfaces. An interface is selected by cursor pick and its definition card is then displayed for the user to modify.

6.13.5 LIST

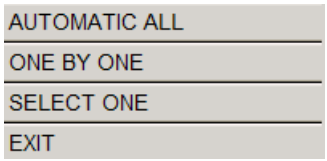
Description: This function lists the defined interfaces in the CONTACT CARD LIST.

6.13.6 RENUMBER

Description: This function renumbers the ID for a selected contact or for all contacts.

Usage:

1. VPG prompts for the option:
   > SELECT OPTIONS



2. If the user enters AUTOMATIC ALL, VPG prompts:
   > ENTER FIRST CONTACT ID. & INC., TYPE 0 TO EXIT

3. If the user selects ONE BY ONE, VPG prompts for a new ID for each contact. The following prompt will be repeated for each contact in the database.
   > ENTER NEW CONTACT ID (OLD= X), TYPE 0 TO EXIT
   - If the user enters nothing, VPG defaults to the original values.

4. If the user enters SELECT ONE, VPG prompts:
   > SELECT ONE CONTACT
   - Once the user selects one contact card, VPG prompts:
     > ENTER NEW CONTACT ID (OLD= X), TYPE 0 TO EXIT

6.13.7 REVERSE NORMAL

Description: This function allows the user to reverse the contact orientation (normal) for all contacts.

Usage:

1. The CONTACT CARD LIST pops up and simultaneously VPG prompts:
   > PICK CONTACT TO REVERSE NORMAL
   • Once the user selects one contact card, VPG prompts:
   > PICK CONTACT PORTION TO REVERSE NORMAL

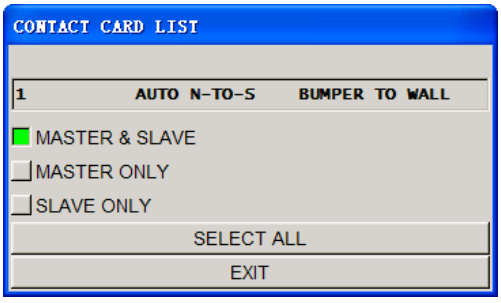
2. After the user selects MASTER (or SLAVE), VPG prompts:
   > SELECT ELEMENTS

6.13.8 SHOW

Description: This function shows the contact area with filled color. Upon selection of the SHOW command, the CONTACT CARD LIST is displayed. When the desired interface is selected, the contact area will be shown with filled color, master is shown as red, slave is shown as blue.

Usage:

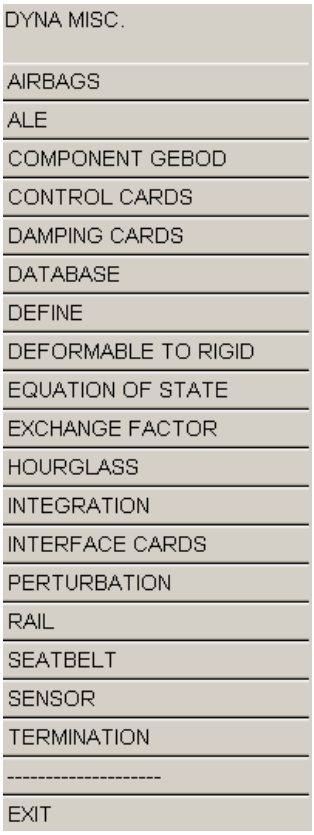
1. VPG prompts:
   > THIS CONTACT TYPE CANNOT BE SHOWN
   • The CONTACT CARD LIST is as following:



Note: The displayed contact may be erased by clicking the CLEAR button in the DISPLAY PARAMETER OPTIONS WINDOW.

6.14 DYNA MISCELLANEOUS

The DYNA MISCELLANEOUS MENU contains distinct LS-DYNA-specific functions. VPG defines these functions in two ways; graphically when defining such things such as a box or vector and with cards based on the definition cards found in the LS-DYNA Keyword User's Manual. An example of a definition card can be found in Section 2.13.



Note: The DYNA MISCELLANEOUS option does not appear in the PRE PROCESSING MENU when the analysis software is NASTRAN.

Note: All of these options share this standard submenu except CONTROL CARDS. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command, this command lists the defined cards in a pop-up window and allows the user to select and modify them.

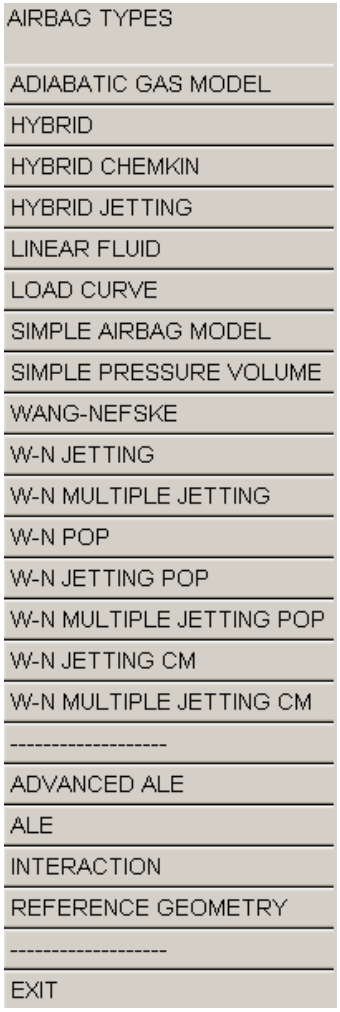
EXIT

6.14.1 AIRBAGS

Description: This function provides various cards for defining the thermodynamic behavior of gas flow into an airbag as well as a reference configuration for the fully inflated bag.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



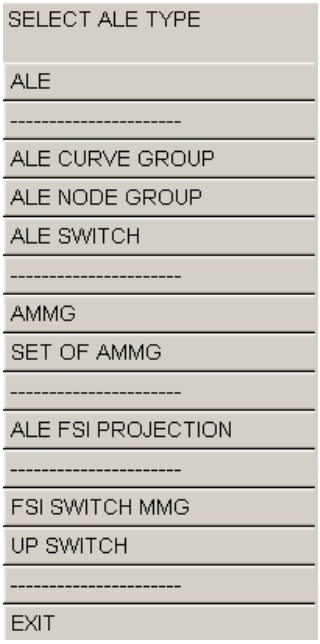
- The preceding commands yield varying definition cards. Refer to the LS-DYNA Keyword User's Manual for further description.

6.14.2 ALE

Description: This function sets the default control parameters for Arbitrary Lagrange-Eulerian and Eulerian calculations.

Usage:

1. VPG prompts:
   > SELECT OPTION



- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

6.14.3 COMPONENT GEBOD

Description: This function sets the default control parameters for component gebod.

Usage:

1. VPG prompts:
   > SELECT OPTION

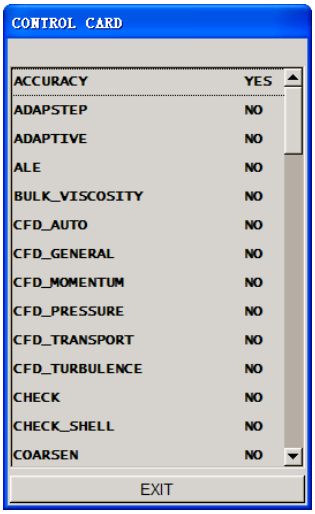
- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

6.14.4 CONTROL CARDS

Description: Control cards are optional default settings.

Usage:

1. VPG prompts:
   > LEFT BUTTON TO DEFINE OR MODIFY; RIGHT BUTTON TO REMOVE
   > SELECT CARD TO CREATE



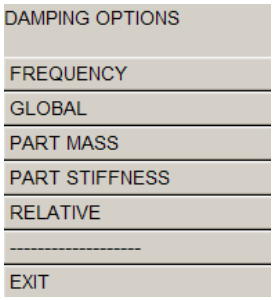
- A **YES** in the right column indicates that this type of control card has been defined.
- A **NO** in the right column indicates that this type of control card has not been defined.

6.14.5 DAMPING CARDS

Description: Damping cards define mass weighted nodal damping that applies globally to the nodes of deformable bodies.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



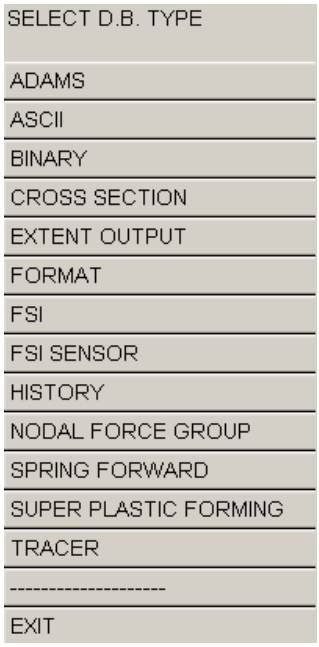
- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

6.14.6 DATABASE

Description: Database definition cards are used to obtain output result files.

Usage:

1. VPG prompts:
   > SELECT DATABASE TYPE



-DATABASE: ADAMS

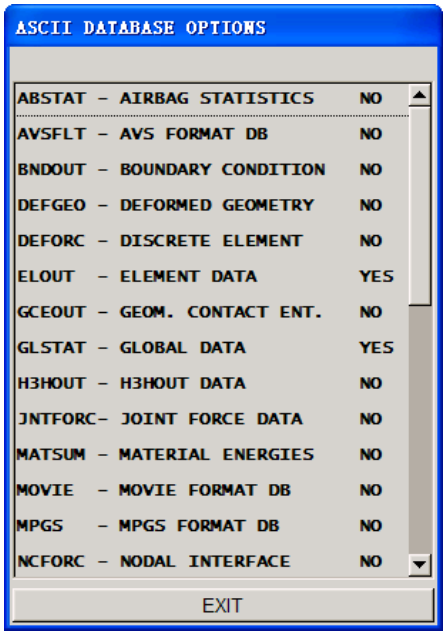
Description: ability to export data in an XML format, which is intended to facilitate sharing of the data with other applications. Imported XML data generated by Adams creates all of the topology of the suspension, the bushings, springs, dampers, rigid links, and beam elements that make up a suspension.

-DATABASE: ASCII

Description: The definition cards in the ASCII subsection specify the ASCII database output files.

Usage:

1. When DEFINE is selected, VPG prompts:
   > SELECT CARD TO DEFINE


ASCII DATABASE OPTIONS
ABSTAT - AIRBAG STATISTICS NO
AVSFLT - AVS FORMAT DB NO
BNDOUT - BOUNDARY CONDITION NO
DEFGEO - DEFORMED GEOMETRY NO
DEFORC - DISCRETE ELEMENT NO
ELOUT - ELEMENT DATA YES
GCEOUT - GEOM. CONTACT ENT. NO
GLSTAT - GLOBAL DATA YES
H3HOUT - H3HOUT DATA NO
JNTFORC- JOINT FORCE DATA NO
MATSUM - MATERIAL ENERGIES NO
MOVIE - MOVIE FORMAT DB NO
MPGS - MPGS FORMAT DB NO
NCFORC - NODAL INTERFACE NO
EXIT

2. The user then selects the type of ASCII card to define. VPG displays the definition card to be edited. The time interval, once entered, will be used as the default value for future cards defined. To edit the time interval for subsequently defined cards the user must click once on the card to turn it on and again to have it displayed.

- A **YES** in the right column of the DATABASE ASCII indicates that this type of card has been defined.
- A **NO** in the right column indicates that this type of card has not been defined.
- Refer to the LS-DYNA User's Manual for further description of the DATABASE ASCII definition cards.

-DATABASE: BINARY

Description: The definition cards in the BINARY subsection specify the BINARY database output files.

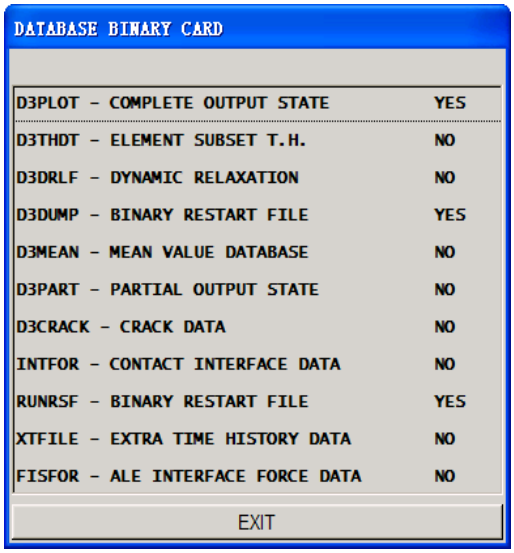
Usage:

1. When DEFINE is selected; VPG displays the DATABASE BINARY CARD and prompts:

   > SELECT CARD TO DEFINE

2. The user then selects the type of BINARY card to define. VPG displays the definition card to be edited. The time interval, once entered, will be used as the default value for future cards defined. To edit the time interval for subsequently defined cards, the user must click once on the card to turn it on and again to have it displayed.

- A YES in the right column of the DATABASE BINARY CARD indicates that this type of card has been defined.
- A NO in the right column indicates that this type of card has not been defined.
- Refer to the LS-DYNA User's Manual for further description of the DATABASE BINARY definition cards.


DATABASE BINARY CARD
D3PLOT - COMPLETE OUTPUT STATE YES
D3THDT - ELEMENT SUBSET T.H. NO
D3DRLF - DYNAMIC RELAXATION NO
D3DUMP - BINARY RESTART FILE YES
D3MEAN - MEAN VALUE DATABASE NO
D3PART - PARTIAL OUTPUT STATE NO
D3CRACK - CRACK DATA NO
INTFOR - CONTACT INTERFACE DATA NO
RUNRSF - BINARY RESTART FILE YES
XTFILE - EXTRA TIME HISTORY DATA NO
FISFOR - ALE INTERFACE FORCE DATA NO
EXIT

-DATABASE: CROSS SECTION

Description: The definition cards in the CROSS SECTION subsection define a cross section for resultant forces written to an ASCII Section Force file.

Usage:

1. When CREATE is selected, VPG prompts:
   > ENTER NEW CROSS SECTION ID (X) OR E TO EXIT

2. After entering new cross section ID, VPG prompts:
   > SELECT CROSS SECTION TYPE
   PLANE (see step 3)
   SET (see step 7)

3. If the user selects PLANE, VPG will first prompt for a part set.
   > SELECT PART SET

   - VPG then prompts for a coordinate system for the cross section.

4. Next, the user will be prompted for the origin of the cross section.

5. VPG then prompts:
   > ENTER LENGTH L (ALONG U) AND M (ALONG V), 0 FOR INFINITE

6. After entering the U, V increments, VPG displays the cross section. The following commands are available for creating and altering new cross sections:
   - **CREATE** defines a new cross section
   - **DELETE** deletes a defined cross section
   - **DISPLAY ON/OFF** toggles the cross section display on/off
   - **MODIFY** redefines a cross section
   - **SHOW SECTION CUT** lists the defined cross sections in the VPG menu
   - **WINDOW**
   - **VIEW**

- **EXIT**

7. If the user selects SET, VPG prompts for a set type. If the type of set selected has not yet been defined, the user will be prompted to create one (see SET MENU, Section 6.6, 6.7). Once the set has been determined, VPG returns to step 1.

Note: Refer to the LS-DYNA User's Manual for further description.

-DATABASE: EXTENT OUTPUT

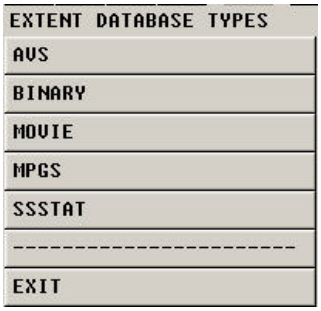
Description: The definition cards in the EXTENT OUTPUT subsection specify the BINARY database output files.

Usage:

1. VPG prompts:
   > SELECT OPTION

   - Refer to the beginning of this section for a description of these commands.

2. When CREATE is selected, VPG prompts:
   > SELECT OPTION



3. When the option is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-DATABASE: FORMAT

Description: The definition cards in the FORMAT subsection specify database format.

Usage:

When DEFINE is selected, the DATABASE FORMAT DEFINITION window appears. User can select result file as DYNA, ANSYS or both

-DATABASE: FSI

Description: This function defines parameters for the fluent data results file.

-DATABASE: FSI SENSOR

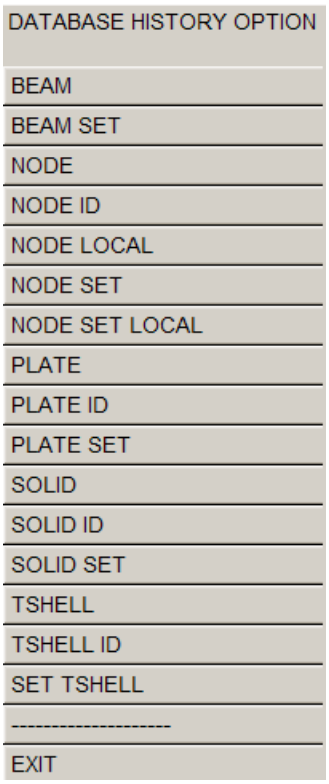
Description: This function defines pressure sensor's locations, which follow the positions of some Lagrangian segments during the simulation.

-DATABASE: HISTORY

Description: The definition cards in the HISTORY subsection specify which nodes or elements are outputted into the binary history file.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

-DATABASE: NODAL FORCE GROUP

Description: The definition cards in the NODAL FORCE GROUP subsection specify a nodal force group for output into ASCII files.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-DATABASE: SPRING FORWARD

Description: The definition card in the SPRING FORWARD subsection creates a spring forward nodal force file.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-DATABASE: SUPER PLASTIC FORMING

Description: The definition card in the SUPER PLASTIC FORMING subsection specifies the output intervals to the superplastic forming output files.

Usage:

1. When DEFINE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

-DATABASE: TRACER

Description: The definition card in the TRACER subsection allows the user to create tracer particles which will save a history of either a material point or a spatial point into an ASCII file; TRHST.

Usage:

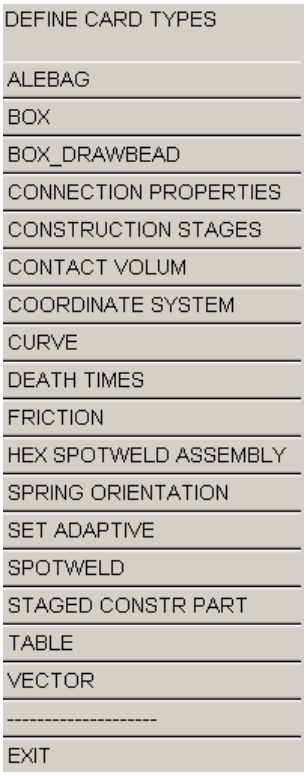
1. When DEFINE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

6.14.7 DEFINE

Description: The DEFINE definition cards allow the user to define boxes, coordinate systems, load curves, tables, and orientation vectors for various uses.

Usage:

1. VPG prompts:
   > SELECT OPTION

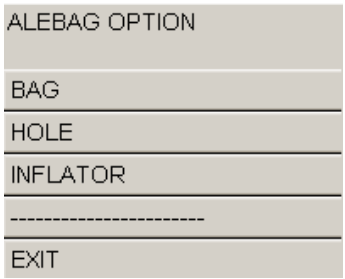


-DEFINE: ALEBAG

Description: Defines information about the Lagrangian airbag structure.

Usage:

1. VPG prompts:
   > SELECT OPTION



2. Select an option, VPG prompts to select following option:

   CREATE
   - This function creates an ALEBAG card

   DELETE
   - This function deletes already created ALEBAG cards

   LIST
   - This function lists already created ALEBAG cards in pop-up window

   MODIFY
   - Allows user to modify already created ALEBAG cards

-DEFINE: BOX

Description: Defines a specific box shaped volume from two corner points specified in global coordinates.

Usage:

1. VPG prompts:
   > ENTER COMMAND
   CREATE
   - This function creates a new box after the user enters a box ID number and defines the two opposing corners using either nodes, points, or xyz coordinates.

   DELETE
   - This function deletes selected boxes. Boxes can be selected via the following commands: CURSOR AT BOX, KEY IN BOX ID, MULTI-POINT REGION, SELECT ALL BOXES, and WINDOW. Boxes that are selected are displayed in gray. When the user exits the menu, they are deleted.

   DISPLAY ON/OFF
   - Toggle switch.

   LIST
   - Lists the box ID numbers in a pop-up window.

   MODIFY
   - Allows the user to modify a selected box with new point, node, or xyz coordinates. This command also displays the UL coordinates, representing the two corners of the box in global coordinates.

   RENUMBER
   - Renumber the selected box ID numbers or all boxes.

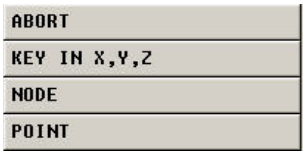
   SHOW
   - Displays the nodes and elements of a box in white. The box may be selected by cursor pick or by box ID.

   VIEW
   - Sends the user to the VIEWING OPTIONS MENU.

   EXIT

2. When CREATE is selected, VPG prompts:
   > ENTER NEW BOX ID. (X) OR E TO EXIT

3. After entering new box ID, VPG prompts:
   > DEFINE FIRST CORNER OF THE BOX

4. After selecting the first corner, VPG prompts:
   > DEFINE SECOND CORNER OF THE BOX

Refer to the LS-DYNA User's Manual for further description.

-DEFINE: BOX DRAWBEAD

Description: Defines a specific box or tube shaped volume around a draw bead.

Usage:

1. VPG prompts:
   CREATE
   - This function creates an BOX DRAWBEAD card

   DELETE
   - This function deletes already created BOX DRAWBEAD cards

   LIST
   - This function lists already created BOX DRAWBEAD cards in pop-up window

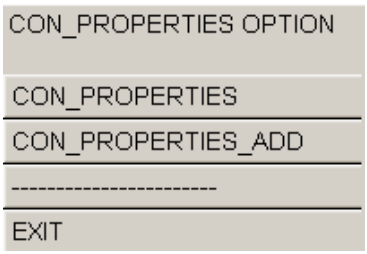
   MODIFY
   - Allows user to modify already created BOX DRAWBEAD cards

-DEFINE: CONNECTION PROPERTIES

Description: Defines failure related parameters for solid element spot weld failure.

Usage:

1. VPG prompts:
   > SELECT OPTION



2. select an option, VPG prompts:
   CREATE
   - This function creates an CONNECTION PROPERTIES card

   DELETE
   - This function deletes already created CONNECTION PROPERTIES cards

   LIST
   - This function lists already created CONNECTION PROPERTIES cards in pop-up window

   MODIFY
   - Allows user to modify already created CONNECTION PROPERTIES cards

-DEFINE: CONSTRUCTION STAGES

Description: Defines times and durations of construction stages.

Usage:

1. VPG prompts:
   CREATE
   - This function creates an CONSTRUCTION STAGES card

   DELETE

- This function deletes already created CONSTRUCTION STAGES cards

LIST

- This function lists already created CONSTRUCTION STAGES cards in pop-up window

MODIFY

- Allows user to modify already created CONSTRUCTION STAGES cards

-DEFINE: CONTACT VOLUME

Description: Defines a rectangular, a cylindrical, or a spherical volume in a local coordinate system.

Usage:

1. VPG prompts:

CREATE

- This function creates an CONTACT VOLUME card

DELETE

- This function deletes already created CONTACT VOLUME cards

LIST

- This function lists already created CONTACT VOLUME cards in pop-up window

MODIFY

- Allows user to modify already created CONTACT VOLUME cards

-DEFINE: COORDINATE SYSTEM

Description: This function allows the user to define a local coordinate system using nodes, points, or vectors.

Usage:

1. When CREATE is selected, VPG prompts:

> SELECT LOCAL SYSTEM TYPE

NODES

- Defines a local coordinate system using three nodes.

SYSTEM

- Defines a local coordinate system from a previously defined coordinate system or key entry.

VECTOR

- Uses a vector to define the coordinate system.

- When using the MODIFY function to modify a coordinate system, VPG displays the CSYS DEFINITION CARD after the DEFINE COORDINATE CARD LIST, allowing the user to key in the exact XYZ coordinates of the system.

-DEFINE: CURVE

Description: This function defines a load curve.

Usage:

1. When CURVE is selected, VPG displays the following options:

CREATE

- When CREATE is selected, VPG prompts:

> ENTER DATA (TIME & VALUE) FOR POINT 1 OR END

- At least two points must be defined to create a curve.
- Once the two points are defined, the curve is displayed.
- Refer to the LS-DYNA User's Manual for further description.

DELETE

- Displays DEFINE CURVE CARD LIST and allows user to select and delete curves.

ERASE UNREF. CURVE

- This function allows the user to DELETE either ALL or SELECTED unreferenced curves in a database.

LIST

- Displays the DEFINE CURVE CARD LIST.

MODIFY

- Displays the DEFINE CURVE CARD LIST and the CURVE DEFINITION CARD once a curve has been selected.

READ CURVE

- Displays the “VPG File Dialog” window and allows the user to read in a previously saved curve (.cur) file.

RENUMBER

- Renumber the selected load curve or all load curves.

SHOW CURVE

- Displays the DEFINE CURVE CARD LIST with all defined curves. By selecting a listed curve and exiting the list, the curve will be displayed.

-DEFINE: DEATH TIME

Description: This function is to dynamically define death time for BOUNDARY PRESCRIBED MOTION based on the location of nodes or rigid bodies.

Usage:

1. VPG prompts:

CREATE

- This function creates an DEATH TIME card

DELETE

- This function deletes already created DEATH TIME cards

LIST

- This function lists already created DEATH TIME cards in pop-up window

MODIFY

- Allows user to modify already created DEATH TIME cards

-DEFINE: FRICTION

Description: This function is to define coefficients between parts for use in the contact options.

Usage:

1. VPG prompts:

CREATE

- This function creates an FRICTION card

DELETE

- This function deletes already created FRICTION cards

LIST

- This function lists already created FRICTION cards in pop-up window

MODIFY

- Allows user to modify already created FRICTION cards

-DEFINE: HEX SPOTWELD ASSEMBLY

Description: This function is to define a list of hexahedral solid elements clusters that make up a single spot weld for computing the force and moment resultants that are written into the SWFORC output file

Usage:

1. VPG prompts:

CREATE
- This function creates an HEX SPOTWELD ASSEMBLY card

DELETE
- This function deletes already created HEX SPOTWELD ASSEMBLY cards

LIST
- This function lists already created HEX SPOTWELD ASSEMBLY cards in pop-up window

MODIFY
- Allows user to modify already created HEX SPOTWELD ASSEMBLY cards

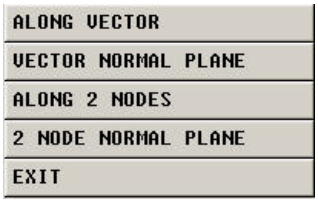
-DEFINE: SPRING ORIENTATION

Description: The definition card in the SPRING ORIENTATION subsection defines orientation vectors for discrete springs and dampers.

Usage:

1. When CREATE is selected, VPG prompts:
   > ENTER NEW VECTOR ID. (X) OR E TO EXIT

2. Once the vector ID has been established, VPG prompts:
   > SELECT SPRING ORIENTATION OPTION



- After defining the vector, it is displayed with its vector number in the display window.

Note: Refer to the LS-DYNA User's Manual for further description.

-DEFINE: SET ADAPTIVE

Description: This function is to control the adaptive refinement level by element or part set.

Usage:

1. VPG prompts:

CREATE
- This function creates an SET ADAPTIVE card

DELETE
- This function deletes already created SET ADAPTIVE cards

LIST
- This function lists already created SET ADAPTIVE cards in pop-up window

MODIFY

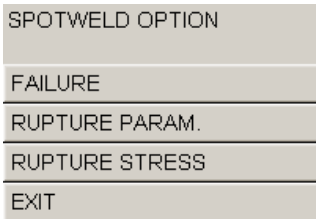
- Allows user to modify already created SET ADAPTIVE cards

-DEFINE: SPOTWELD

Description: This function is to define criteria for spotweld failure, stress or parameter for rupture.

Usage:

1. VPG prompts:
   > SELECT OPTION



2. select an option, VPG prompts:
   DEFINE
   - This function creates an SPOTWELD card

   LIST
   - This function lists already created SPOTWELD cards in pop-up window

   REMOVE
   - This function deletes already created SPOTWELD cards

-DEFINE: STAGED CONSTR PART

Description: This function offers a simple way to define parts that are removed, added and used temporarily during the analysis

Usage:

1. VPG prompts:
   CREATE
   - This function creates an STAGED CONSTR PART card

   DELETE
   - This function deletes already created STAGED CONSTR PART cards

   LIST
   - This function lists already created STAGED CONSTR PART cards in pop-up window

   MODIFY
   - Allows user to modify already created STAGED CONSTR PART cards

-DEFINE: TABLE

Description: The definition card in the TABLE subsection defines a table.

Usage:

1. When **CREATE** is selected, the **TABLE DEFINITION** window appears.

Note: Refer to the LS-DYNA User's Manual for further description.

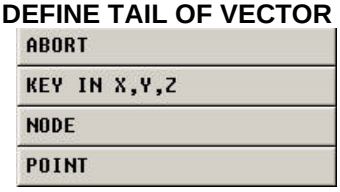
-DEFINE: VECTOR

Description: This function defines a vector using the coordinates of two nodes or points

Usage:

1. When CREATE is selected, VPG prompts:
   > ENTER NEW VECTOR ID. (X) OR E TO EXIT

2. Once the vector ID has been established, VPG prompts:
   > DEFINE TAIL OF VECTOR



3. VPG prompts:
   > DEFINE HEAD OF VECTOR

   - After defining the vector head, the vector is displayed with its vector number in the display window.

Note: Refer to the LS-DYNA User's Manual for further description.

6.14.8 DEFORMABLE TO RIGID

Description: The definition cards of **DEFORMABLE TO RIGID** specify properties of deformable parts. Deformable parts may be switched to rigid at the start of a calculation using the **DEFORM TO RIGID** card. Part switching may be specified on a restart or it may be performed automatically using the **AUTOMATIC** card. The **INERTIA** card sets inertial properties for deformable parts that will be switched to rigid.

Usage:

1. When **CREATE** is selected, VPG prompts to select option.

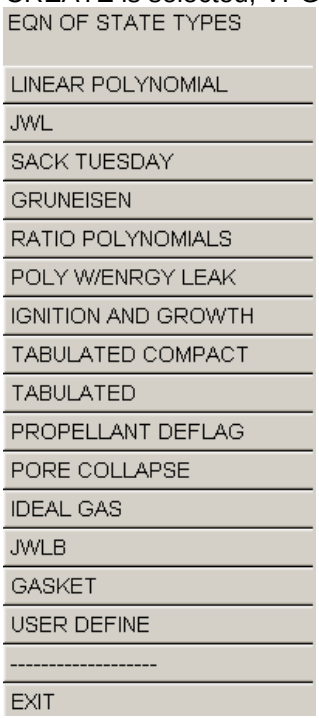
   - The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.9 EQUATION OF STATE

Description: The definition cards of EQUATION OF STATE specify element properties.

Usage:

1. When CREATE is selected, VPG prompts to select option.



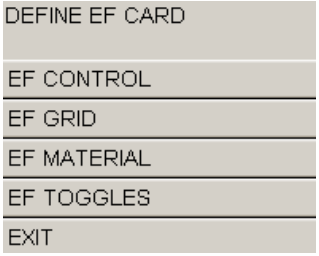
- The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.10 EXCHANGE FACTOR

Description: This function characterizes radiative heat transfer between collections of flat surfaces the union of which is a closed surface (an closure).

Usage:

1. When CREATE is selected, VPG prompts to select option.



- The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.11 HOURGLASS

Description: This function allows the user to define hourglass and bulk viscosity properties.

Usage:

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual for further description.

6.14.12 INTEGRATION

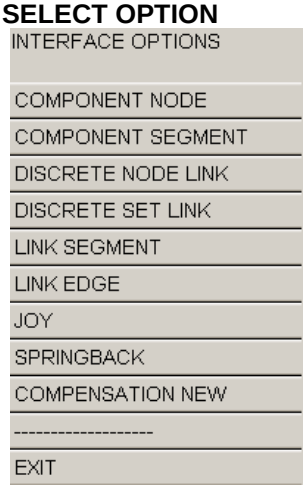
Description: this function is used to define integration parameters for beam and shell elements.

6.14.13 INTERFACE CARDS

Description: This function defines interfaces for linking calculations.

Usage:

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



   - All of the preceding commands yield varying definition cards except for SPRINGBACK. Refer to the LS-DYNA User's Manual for further description of the above commands.

2. Upon selecting SPRINGBACK, VPG displays a pop-up window with the following selections:

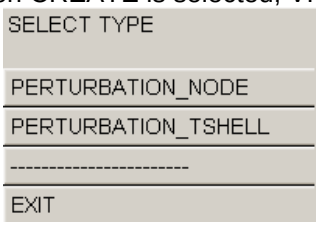
- Once the thickness is determined, VPG displays the appropriate definition card. Refer to the LS-DYNA User's Manual for further description of the above command.

6.14.14 PERTURBATION

Description: This function provides a way of defining deviations from the designed structure such as buckling imperfections.

Usage:

1. When CREATE is selected, VPG prompts to select option.



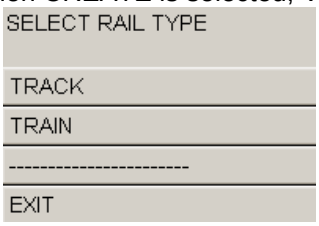
- The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.15 RAIL

Description: This function defines track and train property for railway applications etc.

Usage:

1. When CREATE is selected, VPG prompts to select option.



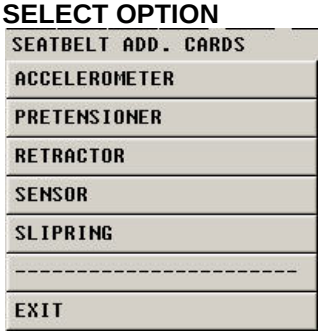
- The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.16 SEATBELT

Description: This function defines a seatbelt.

Usage:

1. When SEATBELT is selected, VPG prompts the following card options:
   > SELECT OPTION



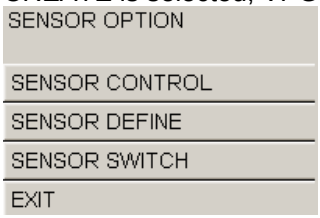
   - Each of these seatbelt functions yields various definition cards. For further description of these cards, refer to the LS-DYNA User's Manual.

6.14.17 SENSOR

Description: This function provides a convenient way of activating and deactivating boundary conditions, airbags, discrete elements, joints, contact, rigid walls, single points constraints, and constrained nodes.

Usage:

1. When CREATE is selected, VPG prompts to select option:



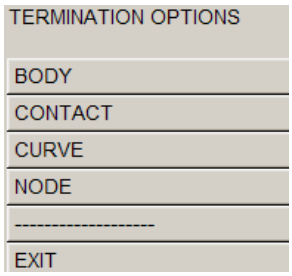
   - The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual for further description.

6.14.18 TERMINATION

Description: This function defines the analysis termination point in reference to a particular node or body. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation.

Usage:

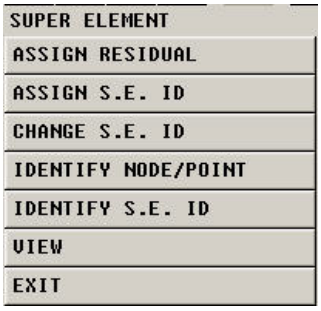
1. When CREATE is selected, VPG prompts:
   > SELECT OPTION



- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual for further description.

6.15 SUPERELEMENT (NASTRAN)

The functions in the SUPERELEMENT (S.E.) menu allow the user to assign and modify SUPERELEMENT I.D. numbers within a VPG database (field 9 of a NASTRAN bulk data GRID card). The following options are available in the SUPERELEMENT menu.



A detailed description of each function is given in the following sections.

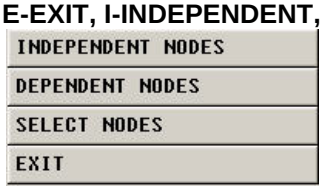
Note: The SUPERELEMENT option does not appear in the PRE PROCESSING MENU when the analysis software is LS-DYNA.

6.15.1 ASSIGN RESIDUAL

Description: This function allows the user to assign a residual I.D. number to selected nodes displayed in the VPG database (SUPERELEMENT I.D. = 0).

Usage:

1. VPG prompts:
   > SELECT AN OPTION FOR RESIDUAL:
   > E-EXIT, I-INDEPENDENT, D-DEPENDENT, S-SELECT



   - If the user selects INDEPENDENT NODES, VPG automatically assigns a residual I.D. number to the independent nodes of all the displayed RBE2 and RBAR elements.
   - If the user enters SELECT NODES, VPG assigns a residual I.D. number to the nodes that are specified by the user.

2. When SELECT NODES is entered, VPG prompts:
   > SELECT NODES
   - The user specifies the nodes that are to receive a residual I.D. number.
   - Once the user selects the residual nodes, DONE or EXIT terminates this function. VPG denotes the residual nodes with a small zero (0) on the screen above the appropriate nodes.

6.15.2 ASSIGN S.E. ID

Description: This function is a one-command operation that assigns a SUPERELEMENT I.D. to all nodes in the entire VPG database (the SUPERELEMENT I.D. is the same as the PID number of the corresponding part).

6.15.3 CHANGE S.E. ID

Description: This function allows the user to change or assign a SUPERELEMENT I.D. to selected nodes.

Usage:

1. VPG prompts:
   > ENTER NEW SUPER EL. I.D OR 0 (ZERO) TO EXIT
   - The user may now enter the desired SUPERELEMENT I.D.

2. VPG prompts:
   > NEW S.E. ID DOES NOT EXIST, OK TO CONTINUE? (Y/N)
   - YES will prompt step 3.
   - NO will exit this function.

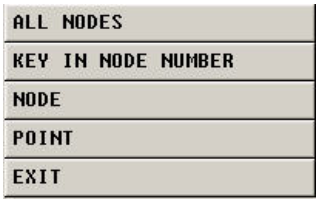
3. VPG prompts:
   > SELECT NODES FOR NEW S.E. ID
   - Once the user selects the S.E. nodes, DONE terminates this function. VPG displays the new S.E. ID numbers on the screen above the appropriate nodes.

6.15.4 IDENTIFY NODE/POINT

Description: This function identifies any point/node and its corresponding global location within the X, Y, Z coordinates.

Usage:

1. VPG prompts:
   > PICK NODES/POINTS



   - If the user selects ALL NODES, VPG displays the numbers of the nodes at their node locations and returns the user to the NODE OPTIONS menu.
   - If the user selects KEY IN NODE NUMBER, VPG displays the corresponding node and echoes the following message on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - If the user selects NODE, VPG displays the number of the node nearest to the cursor and echoes the following message on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - If the user selects POINT, VPG displays the number of the point that is nearest to the cursor and echoes the following message on the command line:
     > POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - EXIT returns the user to the SUPERELEMENT OPTIONS menu.

6.15.5 IDENTIFY S.E. ID

Description: This function allows the user to identify the SUPERELEMENT I.D. number of a selected node(s).

Usage:

1. VPG prompts:
   >SELECT NODES/POINTS

- ALL NODES displays the SUPERELEMENT ID numbers of the nodes.
- The user may select nodes randomly with the cursor to display the node and SUPERELEMENT I.D. numbers.
- EXIT returns the user to the SUPERELEMENT OPTIONS menu.

6.16 MASS MENU (LS-DYNA)

The functions of the MASS MENU are designed to add and modify mass properties in the database. It is necessary sometimes to add masses to represent nonstructural components or to modify masses to change the center of the gravity of the whole body. The functions in this menu are organized as follows:

A detailed description of each function is given in the following sections.

6.16.1 READ FROM FILE

Description: This option allows the user to add mass elements to the database from a file that stores the mass information.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   • The files with the suffix .xml will be listed in the window. The user would select the name of a previously saved file or cancel to exit.

2. If the file does not exist, the prompt reads:
   > CANNOT OPEN THE INPUT FILE

3. VPG reads in the file and returns the user to the MASS MENU.

Note: All mass elements appear as small circles.

6.16.2 WRITE TO FILE

Description: This function allows the user to output all mass information to a file.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   Enter a file name (up to 24 characters).

2. If the file name is a new one, VPG will create it. VPG writes all mass information to the file and returns the user to the MASS MENU.

3. If the file name is old, VPG prompts:
   > WARNING: FILE "*.xml" ALREADY EXISTS, OVERWRITE? (Y/N)
   - **YES**, Overwrite the file with new data.
   - **NO**, Exit and do nothing.

6.16.3 CREATE MASS

Description: This function allows the user add a new mass element to the database.

Usage:

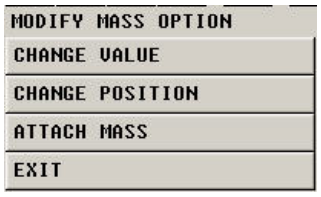
1. VPG prompts:
   > ENTER LUMPED MASS VALUE
   Enter a mass value that the user wants to create.

2. VPG prompts:
   > **PICK NODES/POINTS FOR ELEMENT**
   To create a mass element, the user may select a node, point, or keyboard entry.

EXIT or **ABORT** will exit this function.

6.16.4 MODIFY MASS

Description: The functions of this menu allow the user change the mass value and position. The functions in this menu are organized as follows:



Usage:

1. VPG prompts:
   > SELECT MASS ELEMENT FOR MODIFY

User should select a mass element for modification.

CHANGE VALUE

VPG prompts:
> ENTER ELEMENT VALUE OR 0 TO EXIT
- Enter a new value for the selected mass element. The value of the selected mass will be replaced with the new input value, and VPG returns the user to the MODIFY MASS OPTION menu.

CHANGE POSITION

VPG prompts:
> SELECT NEW POSITION NODE/POINT
- Selects a new node/point for the mass element. The position of the selected mass will be moved to the new position and returns the user to the MODIFY MASS OPTION menu.

ATTACH MASS

VPG prompts:
> ATTACH ALL MASS TO NEAREST NODES? (Y/N/A)
- **YES**, Attach all mass to the nearest nodes.
- **No** and **Abort**, exit and do nothing.

6.16.5 DELETE MASS

Description: This function deletes a selected individual or group of mass elements.

Usage:

1. VPG prompts:
   > ONLY MASS ELEMENTS WILL BE DELETED
   > SELECT ELEMENTS
   User can select mass elements with any methods.
   DONE, The selected mass elements are deleted and return to MASS MENU.
   ABORT, return to the MASS MENU.

6.16.6 IDENTIFY MASS

Description: This function identifies a mass element and provide X,Y, Z geometry, and mass properties, provided in the model unit system.

Usage:

1. VPG prompts:
   > ONLY MASS ELEMENTS WILL BE SELECTED
   > SELECT ELEMENTS
   User selects a mass element and it will display information of this mass.

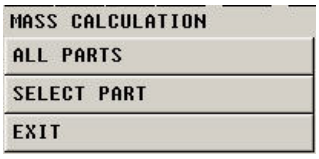
6.16.7 CALCULATE MASS

Description: This function calculates a mass, center of gravity location and inertia properties for selected parts. Parts must have a material property and thickness defined, with a material density specified.

Usage:

1. VPG prompts:
   > ENTER OUTPUT FILE NAME OR [CR] FOR DEFAULT OR "STOP" FOR QUIT
   User inputs a file name:

2. VPG prompts:
   > ENTER COMMAND

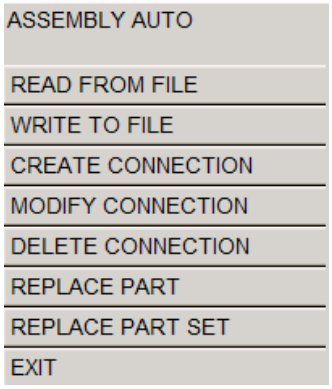


ALL PARTS: Output all mass information of all parts to the input file.
SELECT PART: Output the mass information of the selected part to the input file.
EXIT: Returns the user to the MASS MENU.

6.17 ASSEMBLY (LS-DYNA)

The functions of the ASSEMBLY AUTO menu are designed to add and modify connections in the database. It also includes functions for replacing parts and part sets.

The functions in this menu are organized as follows:



A detailed description of each function is given in the following sections.

6.17.1 READ FROM FILE

Description: This option allows the user to add connections to the database from a file that stores the connection information.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   - The files with the suffix .xml will be listed in the window. The user would select the name of a previously saved file or cancel to exit.

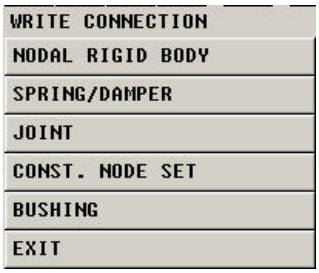
2. If the file does not exist the prompt reads:
   > CANNOT OPEN THE INPUT FILE

3. VPG reads in the file and returns the user to the ASSEMBLY AUTO menu.

6.17.2 WRITE TO FILE

Description: The function allows the user to output connection information to a file. The output format is a simple .xml file, which contains the mass location and properties. This file may be generated outside VPG and imported to create the necessary masses within VPG.

The functions in this menu are organized as follows:



Select one connection type.

Usage:

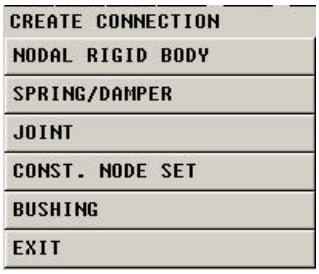
1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   Enter a file name (up to 24 characters).

2. If the file name is a new one, VPG will create it. VPG writes the chosen connection information to the file and returns the user to the WRITE CONNECTION menu.

3. If the file name is old, VPG prompts:
   > WARNING: FILE "*.xml" ALREADY EXISTS, OVERWRITE? (Y/N)
   - **YES**, Overwrite the file with new data.
   - **NO**, Exit and do nothing.

6.17.3 CREATE CONNECTION

Description: This function allows the user to create connections of five types, as following:



Usage:

NODAL RIGID BODY
Refer to section 6.3.4 for detail.

SPRING/DAMPER
Refer to section 6.3.4 for detail.

JOINT
Refer to section 6.3.4 for detail.

CONST. NODE SET
Refer to section 6.3.4 for detail.

BUSHING
1. VPG prompts
   > PICK RIGID BODY TO LOCATE BUSH FIRST NODE

2. After selecting a rigid body, VPG prompts:
> PICK THE FIRST NODE ON RIGID BODY
3. After selecting the first node, VPG prompts:
> PICK RIGID BODY TO LOCATE BUSH SECOND NODE
4. After selecting the second rigid body, VPG prompts:
> PICK THE SECOND NODE TO DEFINE BUSH AXIAL DIRECTION
5. After selecting the second node, VPG prompts:
> PICK THE THIRD NODE TO DEFINE BUSH ORIENTATION
After selecting the third node, material card for bush definition pops up. After clicking OK, the bush definition is finished.

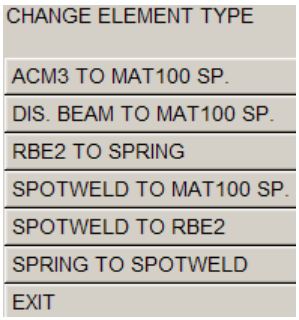
EXIT

6.17.4 MODIFY CONNECTION

Description: This function allows the user to modify connections. Choose one type of them and select a connection. Please refer to section 6.3.11.

Usage:

1. After entering MODIFY CONNECTION, VPG prompts:
> ENTER COMMAND



6.17.5 DELETE CONNECTION

Description: This function allows the user delete connections of the five types. Choose one type of them and select a connection.

Usage:

DONE, The selected connections would be deleted from the database.
ABORT, Do nothing and exit.

6.17.6 REPLACE PART

Description: This function allows the user to select a new part to replace an old part, the nodes index on the old part based on this distance tolerance are switched to mostly close nodes index on the new part.

Usage:

1. VPG prompts:
> SELECT A PART AS TARGET TO BE REPLACED
• Select a part to be replaced by choosing its name or an element of it.

2. After click EXIT, VPG prompts:
   > SELECT A PART TO REPLACE TARGET PART
   • Select a new part by choosing its name or an element of it to replace the part that has been selected.

3. VPG prompts:
   > REPLACE TARGET PART BY THIS PICKED PART? (Y/N)
   • **YES**, The target part will be replaced.

4. VPG prompts:
   > PLEASE INPUT THE TOLERANCE (DEFAULT VALUE: 10)-
   • Key in a tolerance value between the old part and the new part, the nodes index on the old part based on this distance tolerance are switched to mostly close nodes index on the new part

6.17.7 REPLACE PART SET

Description: This function allows the user to select a part set to replace an old part set, the nodes index on the old parts based on this distance tolerance are switched to mostly close nodes index on the new parts.

Usage:

1. VPG prompts:
   > REPLACE PART SET
   > SELECTED FIRST PART SET AS TARGET
   • Select a part set to be replaced from the part set list.

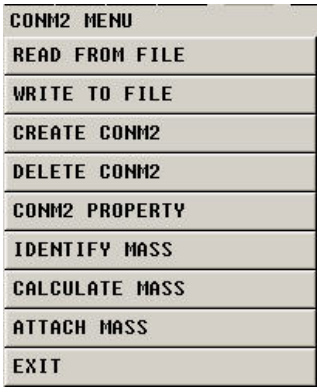
2. VPG prompts:
   > SELECTED THE PART SET TO REPLACE THE TARGET
   • Select another part set from the part set list to replace the part set that has been selected.

3. VPG prompts:
   > REPLACE TARGET PART SET BY THIS PICKED ONE? (Y/N)
   • **YES,** The target part set will be replaced.

4. VPG prompts:
   > PLEASE INPUT THE TOLERANCE VALUE (DEFAULT VALUE: 10)-
   • Key in a tolerance value between the old parts and the new parts, the nodes index on the old parts based on this distance tolerance are switched to mostly close nodes index on the new parts.

6.18 MASS MENU (NASTRAN)

The functions of the MASS MENU are designed to add and modify mass properties in the database. It is necessary sometimes to add masses represent nonstructural components or to modify masses to change the center of the gravity of the whole body. The functions in this menu are organized as follows:



A detailed description of each function is given in the following sections.

6.18.1 READ FROM FILE

Description: This option allows the user to add mass elements to the database from a file that stores the mass information.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   - The files with the suffix .xml will be listed in the window. The user would select the name of a previously saved file or cancel to exit.

2. If the file does not exist the prompt reads:
   > CANNOT OPEN THE INPUT FILE

3. VPG reads in the file and returns the user to the MASS MENU.

Notes: All mass elements appear as small circles.

6.18.2 WRITE TO FILE

Description: This function allows the user to output all mass information to a file.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   Enter a file name (up to 24 characters).

2. If the file name is a new one, VPG will create it. VPG writes all mass information to the file and returns the user to the MASS MENU.

3. If the file name is old, VPG prompts:
   > WARNING: FILE "" ALREADY EXISTS, OVERWRITE? (Y/N)**
   - **YES,** Overwrite the file with new data.
   - **NO,** Exit and do nothing.

6.18.3 CREATE CONM2

Description: This function allows the user create a new mass element to the database. For details, please refer to Section 6.3.

6.18.4 DELETE CONM2

Description: This function deletes a selected individual or group of CONM2 elements.

Usage:

1. VPG prompts:
   > DELETE CONM2
   > SELECT ELEMENTS
   User may select a CONM2 element with any methods.
   DONE: The selected CONM2 element will be deleted, and VPG will exit this function.
   ABORT: Rejects selected CONM2 elements and exists the command.

6.18.5 CONM2 PROPERTY

Description: The details of this function refer to Section 6.12.

Usage:

1. VPG prompts:
   > SELECT AN ELEMENT (C) OR MATERIAL NAME (M) OF A MATERIAL

6.18.6 IDENTIFY MASS

Description: This function identifies a mass element (CONM2) and provide X,Y, Z geometry, and mass properties, provided in the model unit system.

Usage:

1. VPG prompts:
   > SELECT ELEMENT BY CURSOR
   - The user selects a mass element, and VPG will display the information of this mass.
   - **EXIT** to MASS MENU.

6.18.7 CALCULATE MASS

Description: This function calculates a mass, center of gravity location and inertia properties for selected parts. Parts must have a material property and thickness defined, with a material density specified.

Usage:

1. VPG prompts:

> ENTER OUTPUT FILE NAME OR [CR] FOR DEFAULT OR "STOP" FOR QUIT
- User inputs a file name.

2. VPG prompts:
   > ENTER COMMAND

ALL PARTS: Output all mass information of all parts to the input file.
SELECT PART: Output the mass information of the selected part to the input file.
EXIT: Returns the user to MASS MENU.

6.18.8 ATTACH MASS

Description: This function creates RBE (ALL DOFs) elements in the current part and attaches them from the selected CONM2 elements to the nearest nodes in the surrounding parts.

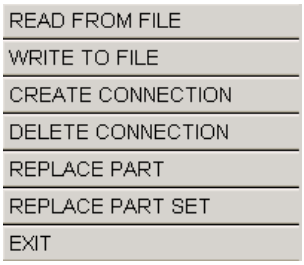
Usage:

1. VPG prompts:
   > ATTACH ALL MASS TO NEAREST NODES? (Y/N/A)
   YES, attach all mass to the nearest nodes.
   NO and **ABORT**, do nothing and return the user to the CONM2 MENU.

6.19 ASSEMBLY (NASTRAN)

The functions of the AUTO ASSEMBLY MENU are designed to add and modify connections in the database. It also includes functions for replacing parts and part sets.

The functions in this menu are organized as follows:



6.19.1 READ FROM FILE

Description: This option allows the user to add connections to the database from a file that stores the connection information.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   - The files with the suffix .xml will be listed in the window. The user would select the name of a previously saved file or cancel to exit.

2. If the file does not exist the prompt reads:
   > CANNOT OPEN THE INPUT FILE

3. VPG reads in the file and returns the user to the AUTO ASSEMBLY menu.

6.19.2 WRITE TO FILE

Description: The function allows the user to output connection information to a file. The functions in this menu are organized as follows:

The user selects one kind of connection.

Usage:

1. VPG prompts:
   > PLEASE INPUT THE FILE NAME
   > ENTER DATABASE FILE NAME
   - Enter a file name (up to 24 characters).

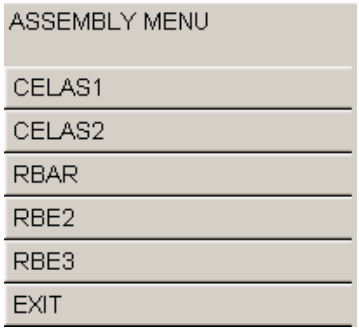
2. If the file name is a new one, VPG will create it. VPG writes the chosen connection information to the file and returns the user to the WRITE CONNECTION menu.

3. If the file name is old, VPG prompts:
   > **WARNING: FILE "**" ALREADY EXISTS, OVERWRITE? (Y/N)**
   - **YES,** Overwrite the file with new data.
   - **NO,** Exit and do nothing.

6.19.3 CREATE CONNECTION

Description: This function allows the user to create connections of five types. For the details of this function, refer to section 6.3.4.

Usage:



6.19.4 DELETE CONNECTION

Description: This function allows the user to delete connections of the five types. Choose one type of them and select a connection.

Usage:

DONE, The selected connections would de deleted from the database.
ABORT, DO nothing and exit.

6.19.5 REPLACE PART

Description: This function allows the user to select a new part to replace an old part, the nodes index on the old part based on this distance tolerance are switched to mostly close nodes index on the new part.

Usage:

1. VPG prompts:
   > **SELECT A PART AS TARGET TO BE REPLACED**
   - Select a part to be replaced by choosing its name or an element of it.

2. After click EXIT, VPG prompts:
   > **SELECT A PART TO REPLACE TARGET PART**
   - Select a new part by choosing its name or an element of it to replace the part that has been selected.

3. VPG prompts:

> REPLACE TARGET PART BY THIS PICKED PART? (Y/N)
- **YES**, The target part will be replaced.

4. VPG prompts:
   > PLEASE INPUT THE TOLERANCE (DEFAULT VALUE: 10)-
   - Key in a tolerance value between the old part and the new part, the nodes index on the old part based on this distance tolerance are switched to mostly close nodes index on the new part

6.19.6 REPLACE PART SET

Description: This function allows the user to select a part set to replace an old part set, the nodes index on the old parts based on this distance tolerance are switched to mostly close nodes index on the new parts.

Usage:

1. VPG prompts:
   > REPLACE PART SET
   > SELECTED FIRST PART SET AS TARGET
   - Select a part set to be replaced from the part set list.

2. VPG prompts:
   > SELECTED THE PART SET TO REPLACE THE TARGET
   - Select another part set from the part set list to replace the part set that has been selected.

3. VPG prompts:
   > REPLACE TARGET PART SET BY THIS PICKED ONE? (Y/N)
   - **YES,** The target part set will be replaced.

4. VPG prompts:
   > PLEASE INPUT THE TOLERANCE VALUE (DEFAULT VALUE: 10)-
   - Key in a tolerance value between the old parts and the new parts, the nodes index on the old parts based on this distance tolerance are switched to mostly close nodes index on the new parts.

Chapter

7

Chapter 7: VPG MODULES

VPG MODULES includes the following sections: ROAD MENU, SUSPENSION MENU, TIRE MENU, SAFETY MENU, ALE-FSI and DROP TEST (DTM) Menu. The functions in these menus allow the user to quickly and easily construct the components required for proving ground simulations, and in the case of the SAFETY MENU, import tools and set boundary conditions for use in vehicle impact and safety simulations.

7.1 ROAD DEFINITION

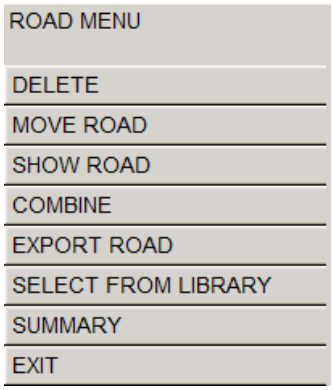
The ROAD DEFINITION menu allows the user to define and save road surfaces. Models may be created using the modeling functions available in eta/VPG, imported using VPG's IGES translator, or selected from VPG's library of road surfaces.

VPG's library currently consists of 11 models. The road surfaces included in this library are digitized models of roads found on the MGA Research proving ground facility in Burlington, WI and make up the majority of the 20,000 mile General Durability Test (see the eta/VPG Applications Manual, Section 1.5).

By default, the road surfaces are defined as rigid bodies with nominal mass. At the user's discretion, the road surface may be modeled using deformable material definitions.

The following functions are included in VPG's ROAD DEFINITION menu:

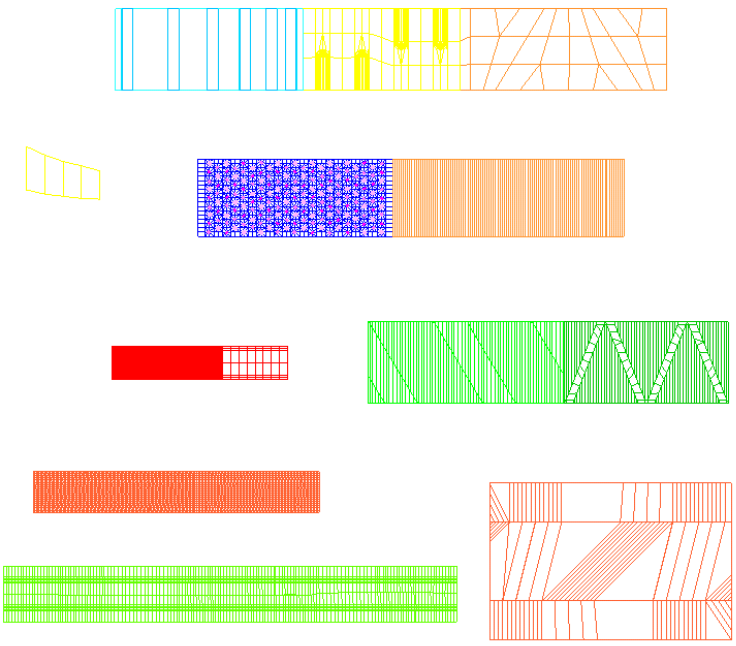
ROAD DEFINITION



A detailed description of each function is given in the following sections.

7.1.1 SHOW ROAD

Description: This option allows the user to look at the 12 types of roads and find which road(s) the user would want to create.



Usage: Select SHOW ROAD from the VPG Road Menu. The road surface models contained in the VPG Road Library will be displayed.

7.1.2 SELECT FROM LIBRARY

Description: This option allows the user to read a Road Surface from the library into the VPG database.

Usage:

1. VPG prompts:
   > SELECT ROAD SURFACE FROM LIBRARY
   > ENTER COMMAND
   ? USER DEFINED
   - Prompts the user for a previously defined road surface library name and allows the user to select saved road surfaces from that library.

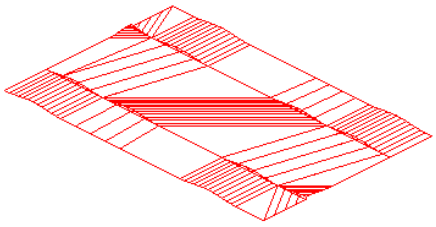
ROAD LIB MENU
USER DEFINED
BODY TWIST LANE
ALTERNATE SURFACE
POTHOLE 1 TRACK
POTHOLE 2 SURFACE
RIPPLE 1 TRACK
RIPPLE 2 TRACK
CHATTERSTRIP
COBBLESTONE TRACK
WASHBOARD 1 SURFACE
PAVE SURFACE
WASHBOARD 2 SURFACE
23-DEG RAMP SURFACE
EXIT

- When one type of road surface library is selected, VPG prompts:

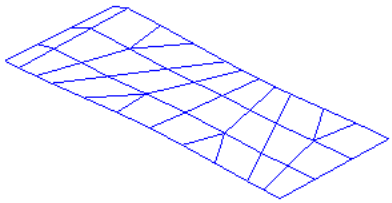
>SELECT C.S TYPE

SELECT C.S. TYPE
FIXED
MOVING
EXIT

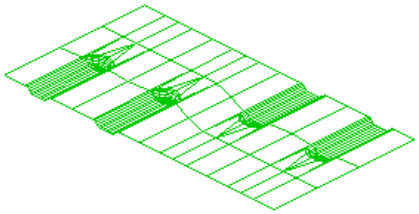
- The desired road surface is displayed on the screen.



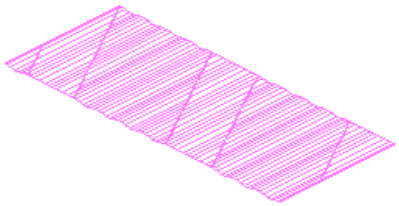
1. BODY TWIST LANE



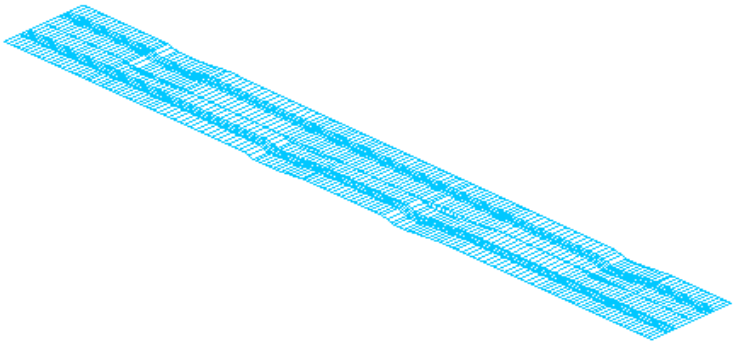
2. ALTERNATE SURFACE



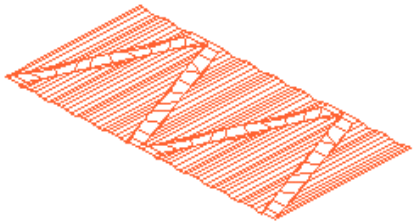
3. POTHOLE TRACK 1



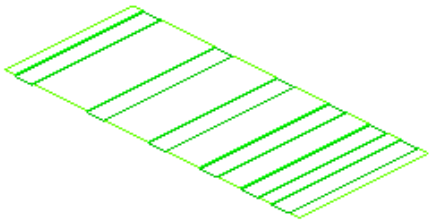
5. RIPPLE TRACK 1



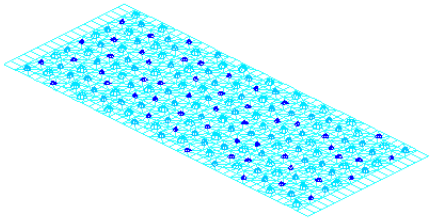
4. POTHHOLE TRACK 2



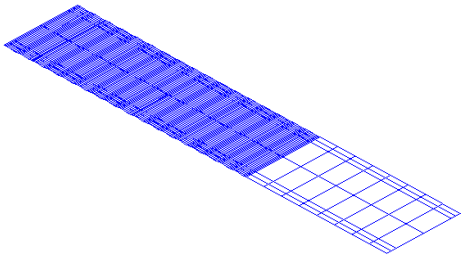
6. RIPPLE TRACK 2



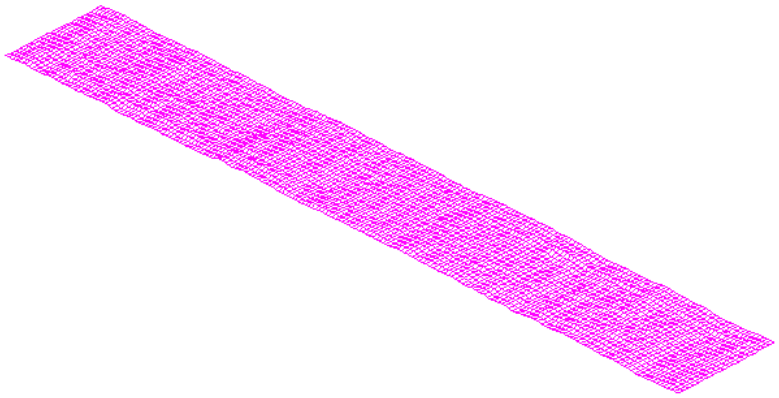
7. CHATTERSTRIP



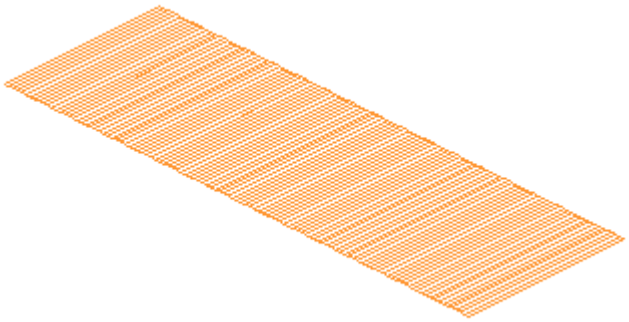
8. COBBLESTONE TRACK



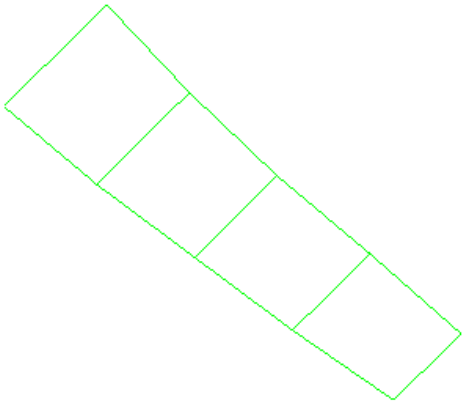
9. WASHBOARD SURFACE 2



10. PAVE SURFACE



11. WASHBOARD SURFACE 2



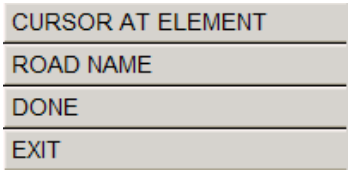
12. 23-DEGREE RAMP SURFACE

7.1.3 DELETE

Description: This option allows the user to delete a defined road surface.

Usage:

1. VPG prompts:
   > **PICK ELEM (C) OR ROAD NAME (R)**
   > **CURSOR PICK AT ELEMENT**



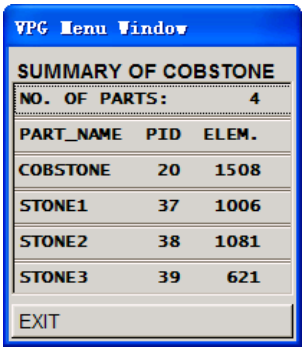
2. Once the road surface is selected, VPG prompts:
   > **XXXX ELEMENTS IN PART XXXX OF ROAD XXXX**
   - The road is deleted.
   - DONE or EXIT returns the user to the ROAD DEFINITION MENU.

7.1.4 SUMMARY

Description: This option gives a summary of a selected road surface, i.e.. Part Name, number of parts, PID, and number of elements.

Usage:

1. VPG prompts:
   > **SELECTED ROAD SURFACE SUMMARY**
   > **SELECT AN ELEMENT ON THE TARGET ROAD SURFACE**
   - Once the desired road surface is selected, its characteristics are displayed in the VPG MENU WINDOW.
2. VPG prompts:
   > **ENTER CR TO EXIT**



7.1.5 MOVE ROAD SURFACE

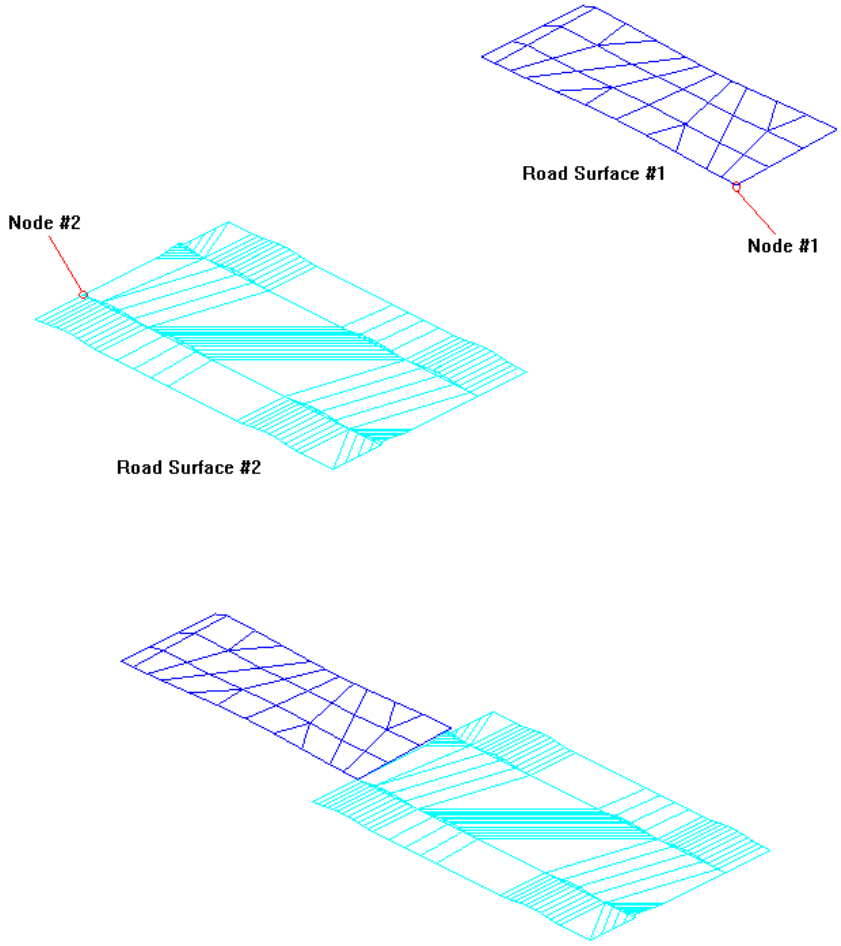
Description: This option allows the user to translate a defined road surface. It is used primarily to move two road surfaces together before combining them into one.

Usage:

1. VPG prompts:
   > **MOVE ROAD SURFACE IN DATABASE**

> **SELECT A NODE IN THE ROAD SURFACE TO BE MOVED**

2. After the first node is selected, VPG prompts:
   > **SELECT A CORRESPONDING NODE FOR THE NEW LOCATION**
   - After the second node is selected, the road surface is moved by aligning the first selected node with the second.



7.1.6 COMBINE

Description: This option allows the user to combine 2 road surfaces into a single, continuous road surface. It is used in conjunction with the MOVE ROAD SURFACE function.

Usage:

1. VPG prompts:
   > **COMBINE ROAD SURFACES IN DATABASE**
   > **SELECT AN ELEMENT ON THE FIRST ROAD SURFACE**
   - The selected road surface is highlighted in white.

2. VPG prompts:
   > **SELECT AN ELEMENT ON THE SECOND ROAD SURFACE**
   - The selected road surface is highlighted in white.

Note: The selected surfaces must be in contact. Otherwise, VPG will prompt:

> BEFORE COMBINING PLEASE MOVE TWO ROADS TOGETHER

3. VPG prompts:
   > ENTER COMBINED ROAD SURFACE NAME
   - The combined road surface will have a new name.

4. Once the name is entered, VPG prompts:
   > ROAD SURFACES COMBINED SUCCESSFULLY

7.1.7 EXPORT ROAD

Description: This option allows the user to save a user-defined or modified road surface to a user-defined road surface library. Multiple road surface libraries can be created with multiple road surfaces in each.

Usage:

1. VPG prompts:
   > SAVE ROAD SURFACE IN LIBRARY
   > ENTER ROAD LIBRARY FILE NAME OR "STOP" TO EXIT
   - This creates the road surface library file or specifies a previously defined library.

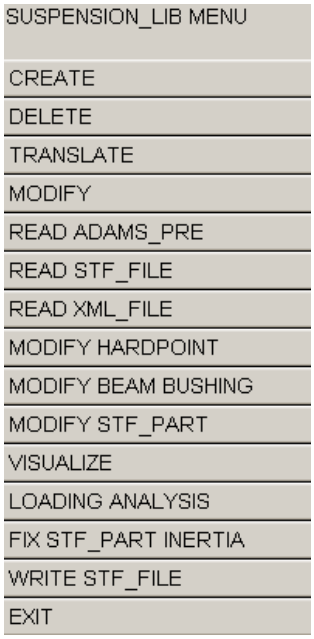
2. VPG prompts:
   > ENTER ROAD SURFACE NAME
   - Once the road surface name is entered, the road surface is saved to the road surface library.

3. To retrieve a saved road surface, access the USER DEFINED key in the SELECT FROM LIBRARY menu. VPG prompts for the library name. Once the library is specified, VPG lists the road surfaces in the MENU WINDOW.

7.2 SUSPENSION MENU

The functions in the **SUSPENSION MENU** allow the user to auto-generate Front and Rear Automotive Suspension beam models from a library of suspension types and easily attach them to the vehicle body model.

The beam chassis/suspension model is commonly used in VPG applications. Although a number of different and detailed finite element suspension models can be used, the simulations will take much more CPU time than they would for simplified beam models. Unless stress/strain history and distribution information are needed for chassis/suspension durability analyses, the beam model is recommended and is both adequate and efficient for most VPG applications such as full vehicle structural durability analysis, road load prediction, full vehicle system NVH analysis, and ride and handling analysis. For more information on suspension definition, see the eta/VPG Applications Manual, Section 1.3. Currently, two Front and eight Rear Suspension types can be auto-generated.



A detailed description of each function is given in the following sections.

7.2.1 CREATE

Description: This option allows the user to define and auto-generate suspension models.

Usage:

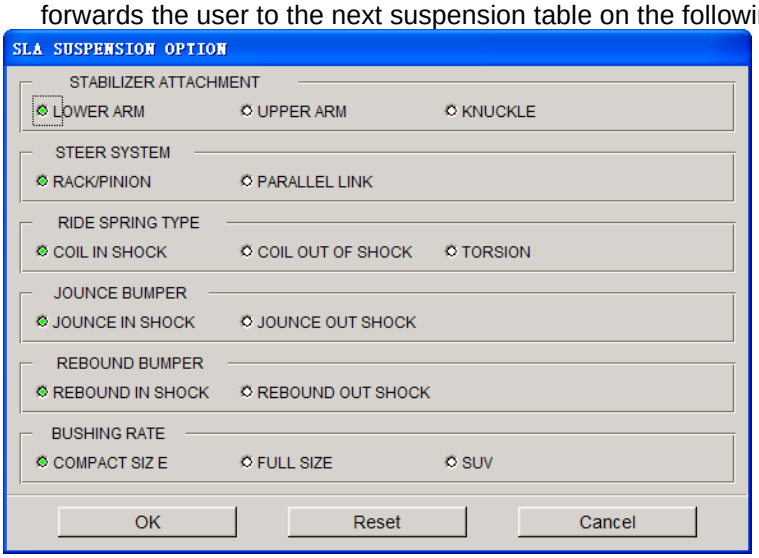
1. VPG prompts:
   > GENERATE SUSPENSION MODEL
   > ENTER COMMAND

SUSPENSION TYPE
FRONT SUSPENSION
1 SHORT LONG ARM
2 MACPHERSON A-ARM

REAR SUSPENSION
3 MACPHERSON H-ARM
4 SOLID AXLE
5 FIVE LINK
6 QUADRA LINK
7 TRAILING ARM
8 HOTCHKISS
9 TWIST BEAM
10 HONDA 5-LINK

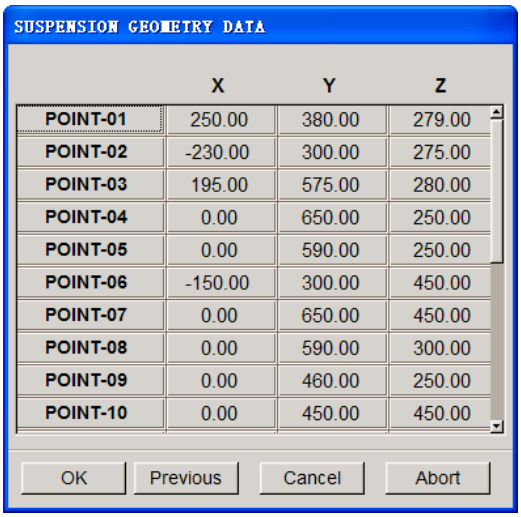
EX-EXIT

2. Depending on the suspension type, VPG will display one or both of the following two tables. The first table displays suspension specific options such as stabilizers, springs, and attachments, and is only necessary for some suspension types. VPG prompts:
   > **MODIFY SUSPENSION OPTIONS**
   - When the user has finished altering the data and has selected OK, VPG forwards the user to the next suspension table on the following page.



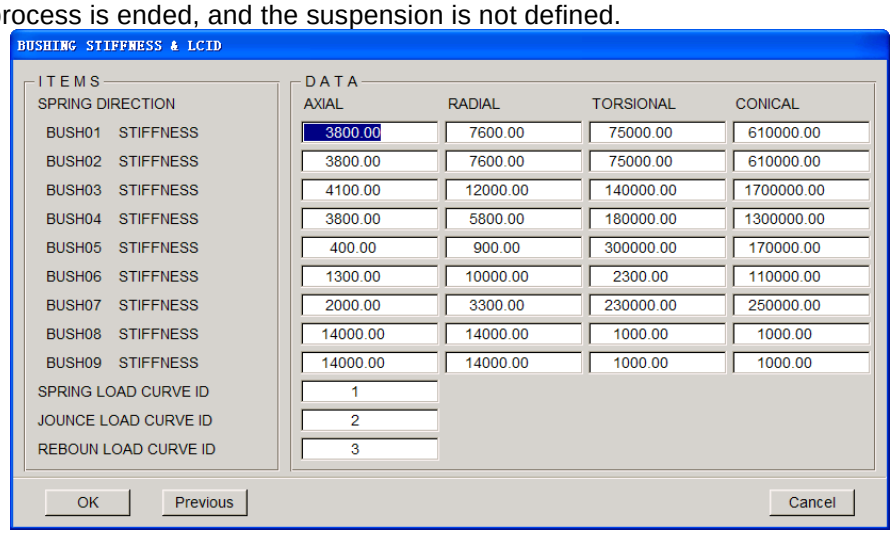
3. VPG prompts:
   > **MODIFY GEOMETRY DATA**
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > **ACCEPT THESE GEOMETRY DATA. (Y/N)**
   - YES forwards the user to the next suspension table (step 4).
   - NO returns the user to the SUSPENSION GEOMETRY table.



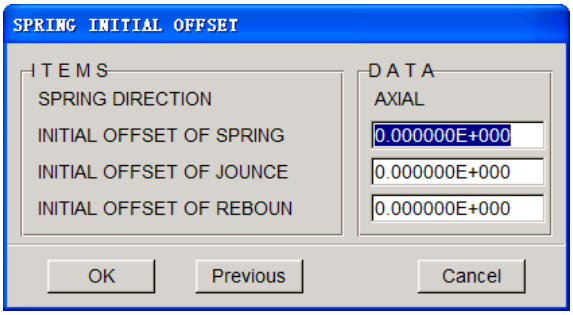
4. VPG displays the following pop-up window with suspension stiffness data, and VPG prompts:
   - **> MODIFY BUSHING STIFFNESS DATA**
   - **> SELECT AN ITEM TO EDIT**
     - When the user has finished altering the data and has selected OK, VPG prompts:
   - **> ACCEPT STIFFNESS DATA. (Y/N)**
     - YES forwards the user to the next suspension table (step 5).
     - NO returns the user to the SUSPENSION BUSHING STIFFNESS table.

   Note: If the user selects cancel at any time during the suspension creation process, the process is ended, and the suspension is not defined.



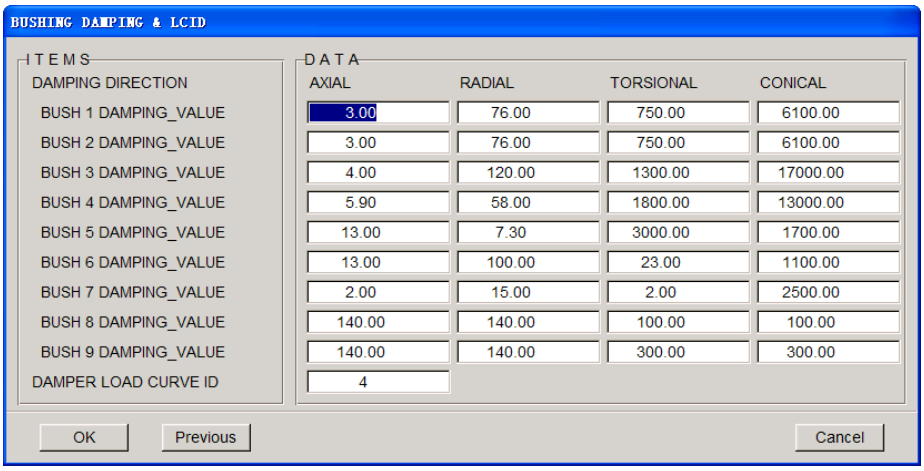
5. VPG displays the following pop-up window with spring damper rebound initial offset data, and VPG prompts:
   - **> MODIFY INITIAL OFFSET**
   - **> SELECT AN ITEM TO EDIT**
     - When the user has finished altering the data and has selected OK, VPG prompts:
   - **> ACCEPT INITIAL OFFSET. (Y/N)**
     - YES forwards the user to the next suspension table (step 6).

- NO returns the user to the BUSHING STIFFNESS table.



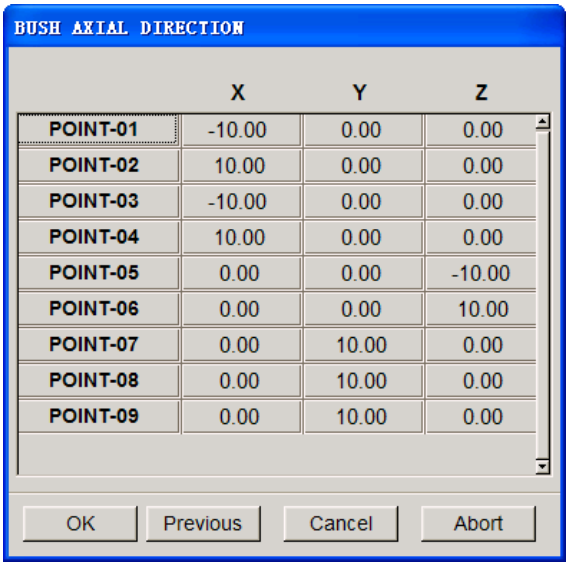
6. VPG displays the following pop-up window with damping coefficient data, and VPG prompts:
   > **MODIFY DAMPING COEFFICIENT DATA**
   > **SELECT AN ITEM TO EDIT**
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > **ACCEPT DAMPING COEFFICIENT DATA. (Y/N)**
   - YES forwards the user to the next suspension table (step 7).
   - NO returns the user to the SPRING INITIAL OFFSET table.



7. VPG displays a pop-up window with bush axial data. VPG prompts:
   > **MODIFY EXTRA NODE COORDINATES**
   > **SELECT AN ITEM TO EDIT**
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > **ACCEPT EXTRA NODE COORDINATES. (Y/N)**
   - NO returns the user to the bush axial data table.
   - If YES is entered VPG prompts:

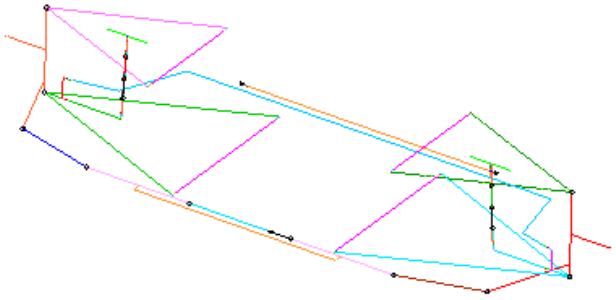


> **USE DEFAULT MASS, CG. & INERTIA MOMENT. (Y/N)**
- YES completes the suspension.

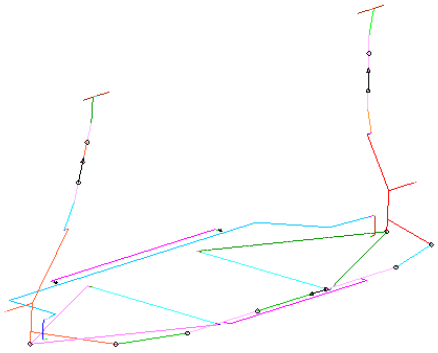
8. If NO is entered VPG prompts the user to select the parts to modify:
> **PICK PARTS TO MODIFY**

Once the parts are selected and the user has exited the list, VPG prompts the user:
> **MODIFY MASS A CG. CENTER**
> **MODIFY INERTIA MOMENT**
- VPG displays a pop-up window with the parts' properties, allowing the user to alter them.

Front Suspension Library

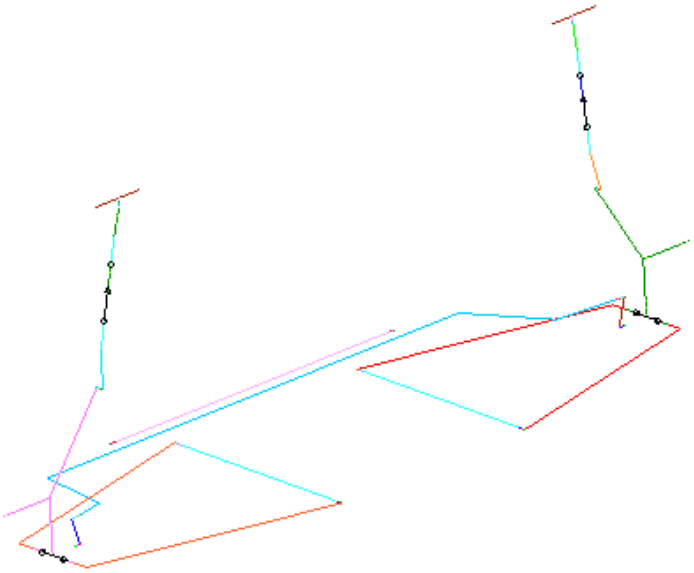


SHORT-LONG A-ARM SUSPENSION

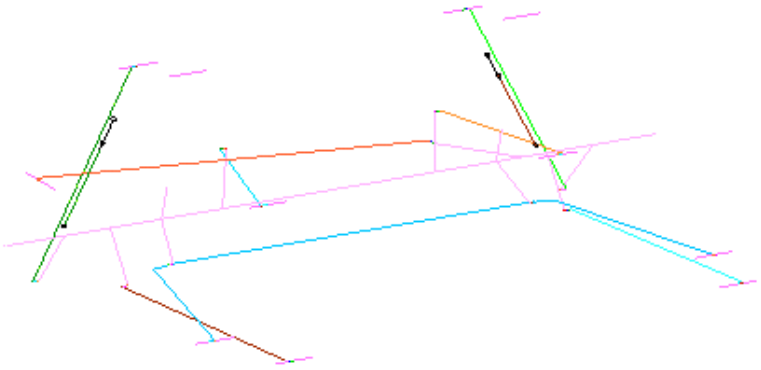


MCPHERSON H-ARM FRONT SUSPENSION

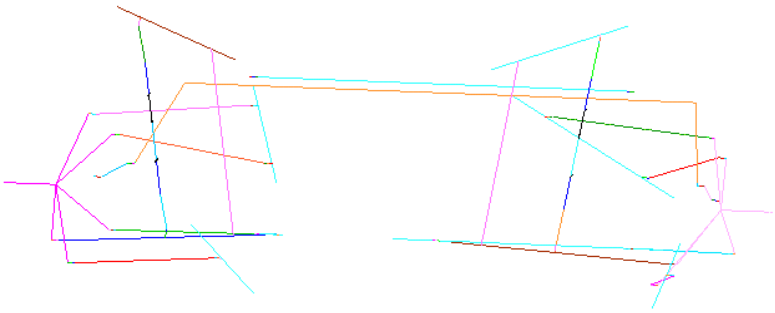
Rear Suspension Library



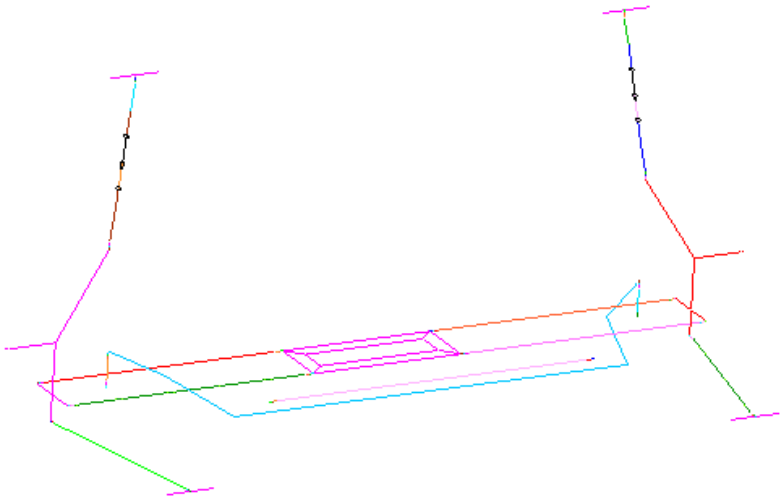
MCPHERSON H-ARM REAR SUSPENSION



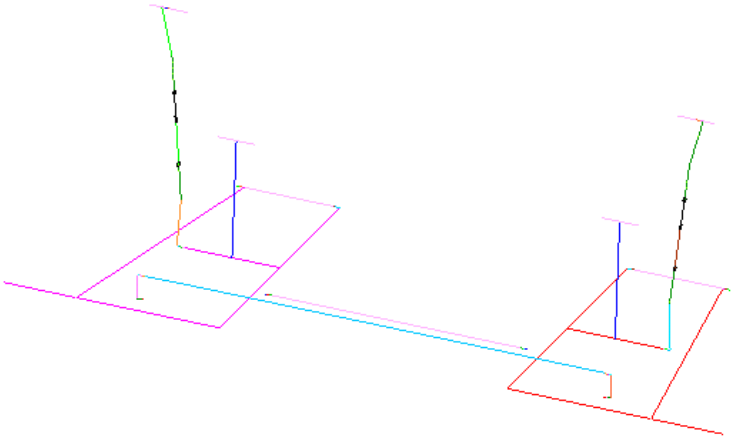
TWIST BEAM REAR SUSPENSION



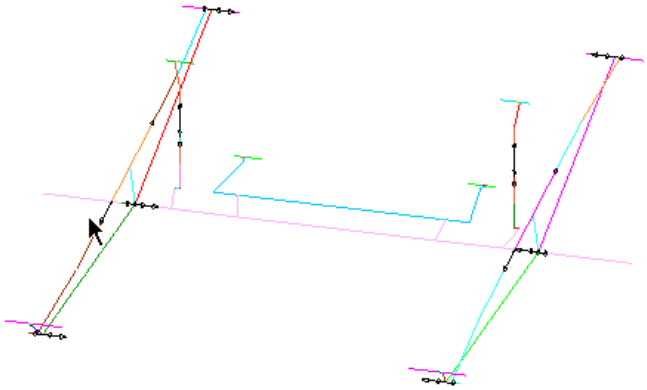
5 -LINK REAR SUSPENSION



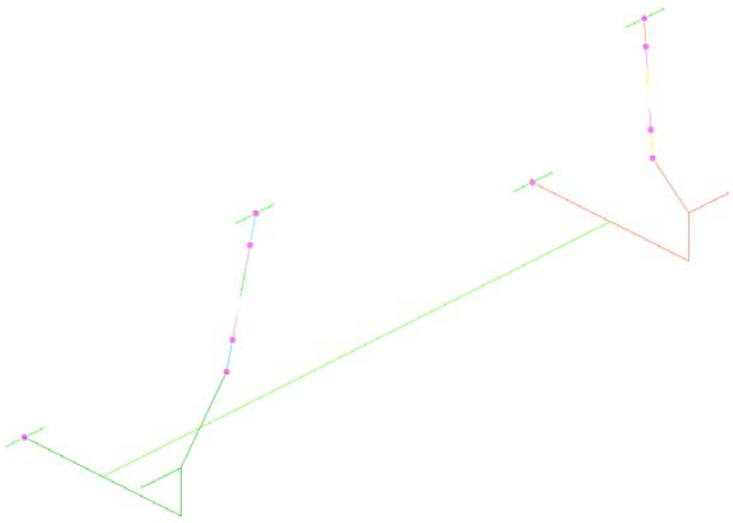
QUADRA LINK REAR SUSPENSION



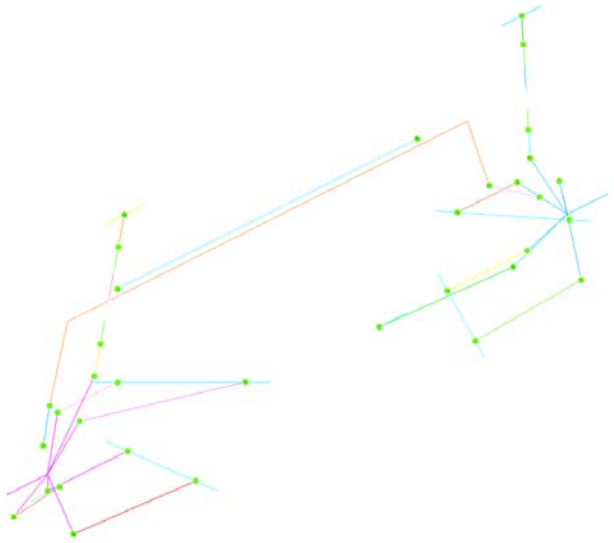
TRAILING ARM REAR SUSPENSION



HOTCHKISS REAR SUSPENSION



TWIST BEAM



HONDA 5_LINK

7.2.2 DELETE

Description: This option allows the user to delete defined suspension models from the VPG database.

Usage:

1. VPG prompts:
   > SELECT SUSPENSION FROM SUSPENSION LIST OR SCREEN
   - Once an element from the suspension is selected, VPG prompts:

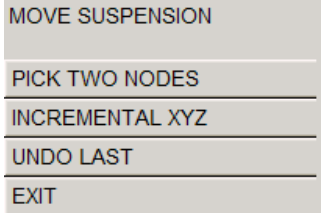
   > OK TO DELETE THIS SUSPENSION. (Y/N)
   - YES deletes the suspension model and returns the user to the SUSPENSION MENU.
   - NO returns the user to the SUSPENSION MENU.

7.2.3 TRANSLATE

Description: This option allows the user to translate auto-generated suspension models. Its purpose is to allow the user to quickly align the auto-generated suspension model with the F.E. body model for attachment.

Usage:

1. VPG prompts:
   > PICK AN ELEMENT TO TRANSLATE SUSPENSION



   - Once the two locations have been determined, the suspension is moved with the second coordinate now occupying the first.

7.2.4 MODIFY

Description: This option allows the user to modify defined suspension models from the VPG database.

Usage:

1. VPG prompts:
   > PICK AN ELEMENT TO MODIFY SUSPENSION
   - Once an element from the suspension is selected, VPG prompts:

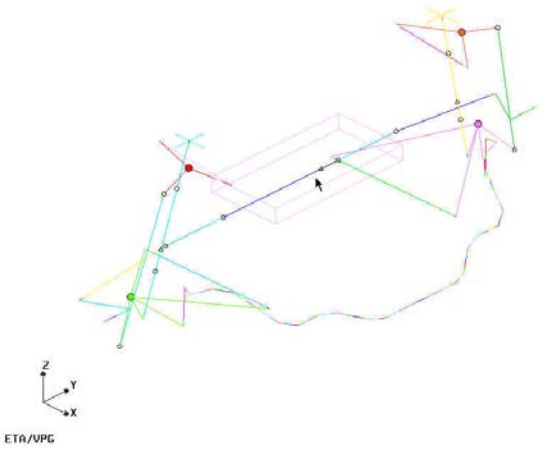
   > OK TO MODIFY THIS SUSPENSION? (Y/N)
   - **YES** Modify, step by step, the suspension model like in the create method. The user returns to the SUSPENSION MENU.
   - **NO** Returns the user to the SUSPENSION MENU.

7.2.5 READ ADAMS PRE

Description: This option allows the user to read in 10 types of Adams Pre format suspension files to the VPG database.

Usage:

1. VPG prompts:
   > READ ADAMS_PRE MENU
   - The READ ADAMS_PRE FILE MENU window is displayed for user to OPEN a **.**fst*/*.rst database file.



ADAMS-PRE SUSPENSION TYPES SUPPORTED ARE:

FRONT SUSPENSION	SLA Front Suspension
	McPherson Strut Front Suspension
	SLA Multilink Front Suspension
REAR SUSPENSION	Hotchkiss Rear Suspension
	Trailing Arm Sla Independent Rear Suspension
	3 Link Suspension
	4 Link Suspension
	Mazda J56a_Rear Suspension
	Quad-Link Uca Rear Suspension

	Quad-Link Struct Rear Suspension

7.2.6 READ STF_FILE

Description: This function reads in a STF suspension.

7.2.7 READ XML_FILE

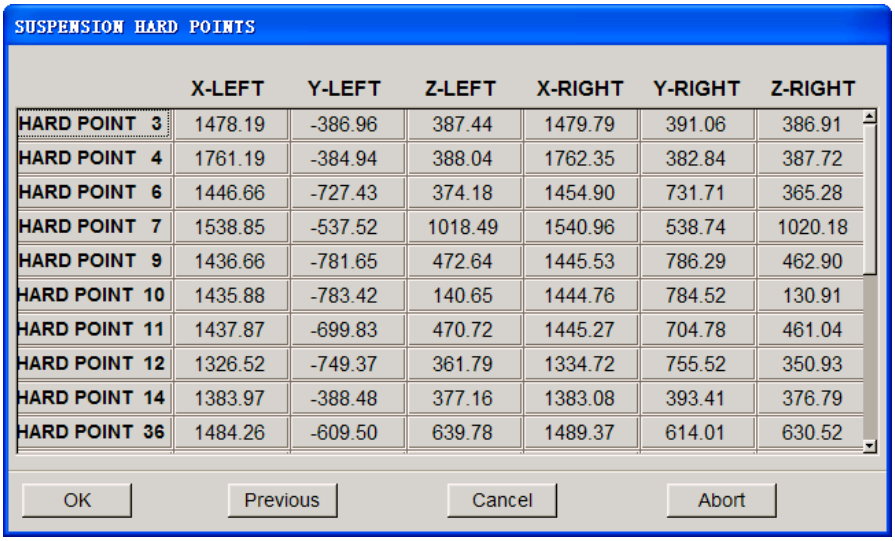
Description: This function reads in a XML suspension.

7.2.8 MODIFY HARDPOINT

Description: When the user has read in an Adams Pre file, this option allows the user to modify defined suspension geometry definitions from within the VPG database.

Usage:

1. VPG prompts:
   > PICK AN ELEMENT TO MODIFY SUSPENSION HARDPOINT
   - The MODIFY HARDPOINT window is displayed for user to modify hard point X, Y, Z positions.



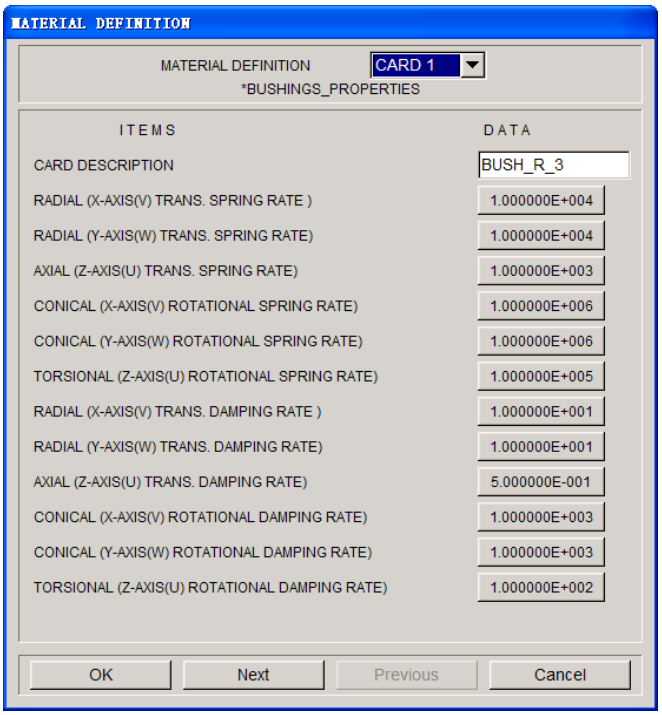
When the user has finished altering the data, select OK.

7.2.9 MODIFY BUSHING

Description: This option allows the user to modify defined suspension models' bushing definition from the VPG database.

Usage:

1. VPG prompts:
   > PICK A BUSHING TO MODIFY SUSPENSION
   - Once a bushing from the suspension is selected, the definition window is displayed as shown.



- When the user has finished altering the data, select OK to accept this change or select cancel to quit.

7.2.10 MODIFY STF_PART

Description: This option allows the user to modify defined suspension models' STF part definition from the VPG database.

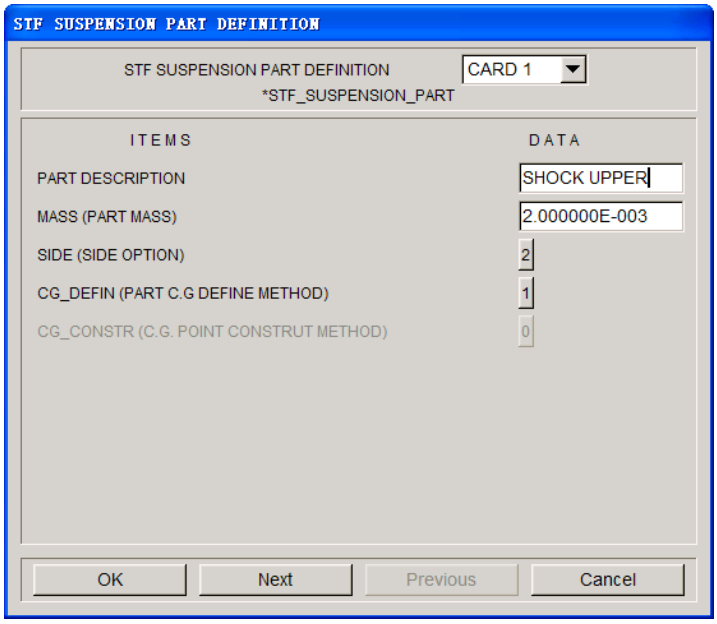
Usage:

1. VPG prompts:

> MODIFY STF_PART

> SELECT PART BY SELECTING AN ELEMENT THE PART OR SELECTING FROM THE PART LIST

- Once a STF part from the suspension is selected, the definition window is displayed as shown.



- When the user has finished altering the data, select OK to accept this change or select cancel to quit.

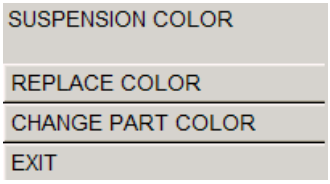
7.2.11 VISUALIZE SUSPENSION

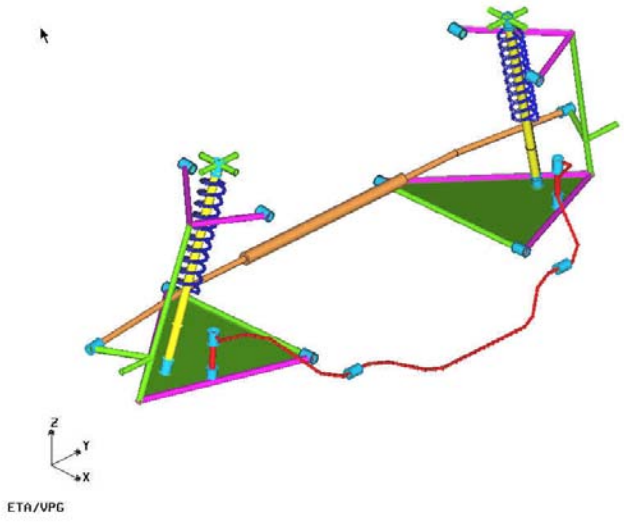
Description: This option allows the user to visualize a suspension and change part color. VPG will create temporary surfaces that can be shaded for visualization purposes. These temporary surfaces are deleted upon exiting this function.

The user may change the suspension component colors by selecting CHANGE PART COLOR from the menu. The user then selects the part to modify from a screen selection or from the part listing. The new color is then selected from the color bar displayed in the Display Window. This function is terminated by selecting the EXIT command.

Usage:

1. VPG prompts:





7.2.12 LOADING ANALYSIS

Description: This option allows the user to define an LS-DYNA file to analyze STF or XML suspension motion.

CREATE
DELETE
LIST
MODIFY
READ BLC STF FILE
G VALUE(SPRUNG MASS)
g VALUE(UNSPRUNG MASS)

EXIT

7.2.12.1 CREATE

Description: This option allows the user to create boundary condition, load, and constraint for model.

Usage:

1. VPG prompts:

BOUNDARY HARDPOINT
BOUNDARY PART
LOAD HARDPOINT
LOAD PART
CONSTRAINED PART
CONTROL CARD

EXIT

BOUNDARY HARDPOINT

- User can define boundary conditions such as velocity, acceleration and displacement on hard point of suspension.

BOUNDARY PART
- User can define boundary conditions such as velocity, acceleration and displacement on part of suspension.

LOAD HARDPOINT
- User can add load on hard point of suspension, load type can be defined as constant, ramp up, sinusoidal or by curve.

LOAD PART
- User can add load on part of suspension, load type can be defined as constant, ramp up, sinusoidal or by curve.

CONSTRAINED PART
- User can define constrain between master part and slave part.

CONTROL CARD
- User can define analysis parameter such as termination time and output time step etc.

7.2.12.2 DELETE

Description: This option allows the user to delete already defined boundary condition, load, and constraint in suspension model.

7.2.12.3 LIST

Description: This option lists already defined boundary condition, load, and constraint in suspension model.

7.2.12.4 MODIFY

Description: This option allows the user to modify already defined boundary condition, load, and constraint in suspension model.

7.2.12.5 READ BLC STF FILE

Description: This option allows the user to read in a BLC STF file.

7.2.12.6 G VALUE(SPRUNG MASS)

Description: This option allows the user to define sprung mass.

7.2.12.7 g VALUE(UNSPRUNG MASS)

Description: This option allows the user to define unsprung mass

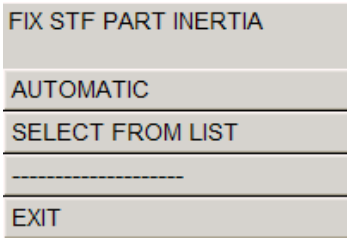
7.2.13 FIX STF_PART INERTIA

Description: This function removes inertia of selected parts from suspension model.

Usage:

1. VPG prompts:

> SELECT OPTION



AUTOMATIC
- All parts inertia in suspension model are removed.

SELECT FROM LIST
- A part list pops up, then inertia of selected part will be removed.

EXIT
- Exit this function without removing inertia.

7.2.14 WRITE STF_FILE

Description: This function export suspension models

Usage:

1. VPG prompts:

> ENTER DATABASE FILE NAME
- Entering file name and click OK, STF_FILE is exported.

7.3 TIRE MENU

The tire model is one of the most important components in a VPG analysis. Although modern pneumatic tires have very complex structures, the VPG/tire model concentrates only on the most important global mechanical properties of the tire. These properties ensure the proper transfer of forces and displacements from the tire/road contact interface to the vehicle chassis/suspension and body structure.

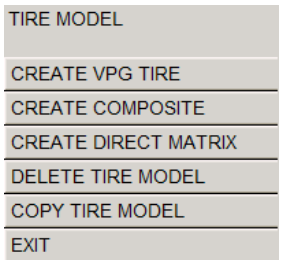
A key advantage of this tire model is its simplicity, giving CPU economy. The tire has the ability to interact with three dimensional road surfaces in a manner consistent with actual tires. This allows for the prediction of lateral and longitudinal force within the vehicle suspension with greater accuracy.

VPG's tire generator creates tires with the following characteristics and properties:
The tire's tread and chafer are Mooney-Rivlin type rubber materials modeled with eight-node solid elements. The tire's top ply and sidewall are modeled as shell elements with elastic material. The wheel is made of rigid shell elements.

Tire inflation pressure is defined according to the formula: $pV/T = \text{constant}$ where p is pressure; V is the volume confined within the top ply, side wall, and the wheel hub; and T is a temperature assumed to be constant during the simulated testing process.

The finished VPG/Tire model contains a total of 2441 nodes and 2264 elements (including 1304 shell elements and 960 solid elements).

The **TIRE MENU** allows the user to generate tire models with user-defined characteristics.



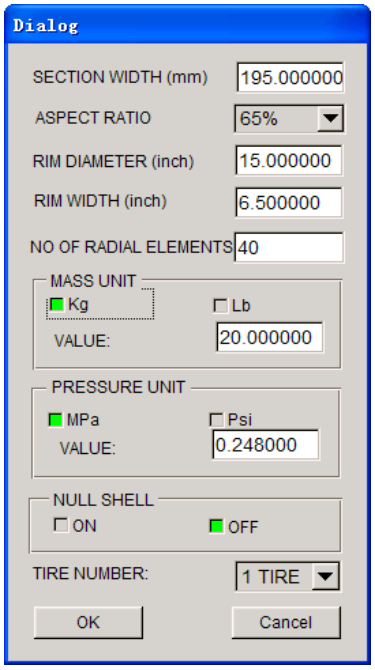
A detailed description of each function is given in the following sections.

7.3.1 CREATE VPG TIRE

Description: This option allows the user to generate VPG tire models based on user-defined parameters.

Usage:

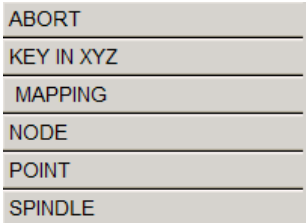
1. VPG displays a pop-up window with tire parameter data and prompts:
   > CREATE TIRE PROFILE

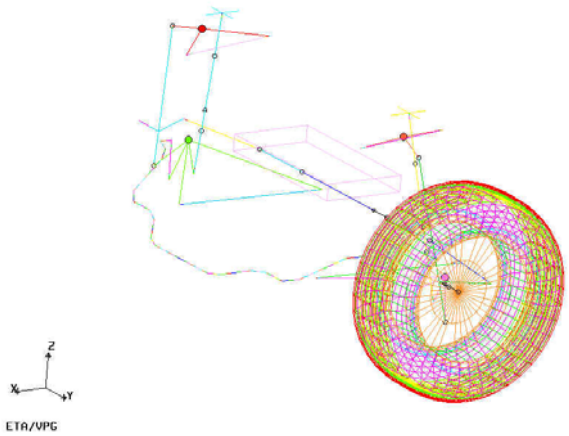


2. When the user has exited the tire parameter window, VPG displays the tire cross-section along with mass and pressure values and prompts:

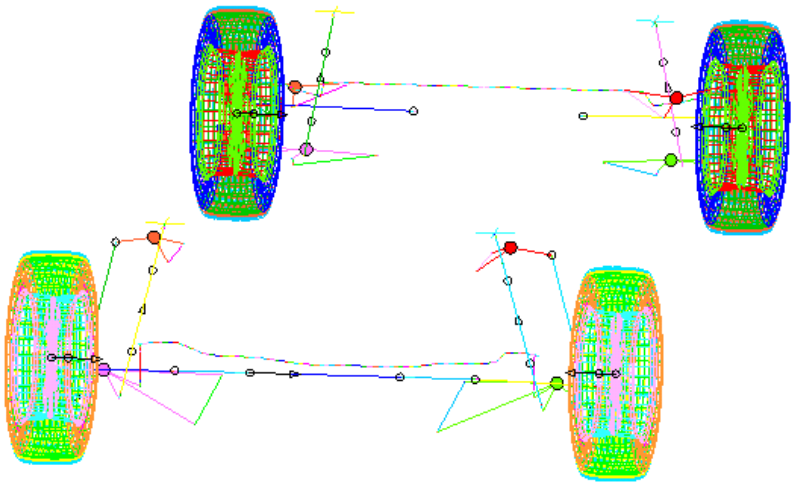
   > PROCESSING ELEMENTS AND NODES
   > DO YOU ACCEPT THIS TIRE PROFILE. (Y/N/ABORT)
   a) NO returns the user to the tire parameter window.

3. If the user accepts the tire profile, VPG prompts:
   > DEFINE WHEEL CENTER LOCATION FOR TIRES





Note: For tire tuning procedures, see the VPG/Structure Tutorial. sections 1.5 and 2.4



7.3.2 DELETE TIRE MODEL

Description: This option allows the user to delete the currently defined tire models.

Usage:

1. VPG prompts:
   > SELECT TIRE FROM TIRE LIST OR SCREEN

2. Once the tire is selected, it is highlighted and VPG prompts:
   > OK TO DELETE THIS TIRE? (Y/N/ABORT)
   - If YES is selected, the tire will be deleted.

7.3.3 COPY TIRE MODEL

Description: This function allows the user to copy a tire model and its characteristics to a specific location.

Usage:

1. VPG prompts:
   > CHOOSE A TIRE FOR COPYING BY PICKING AN ELEMENT

2. After selecting the tire, VPG prompts:
   > ACCEPT THE SELECTED TIRE? (Y/N/ABORT)

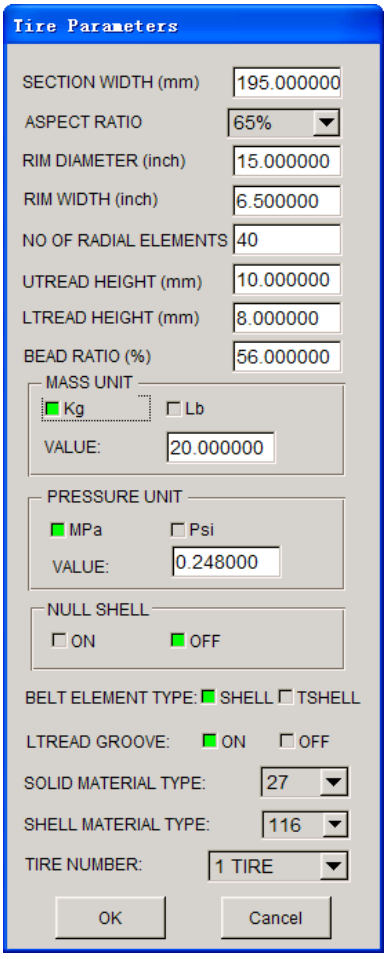
3. The user is then prompted for the location of the new tire. Once entered, the tire is copied to that location.

7.3.4 CREATE COMPOSITE

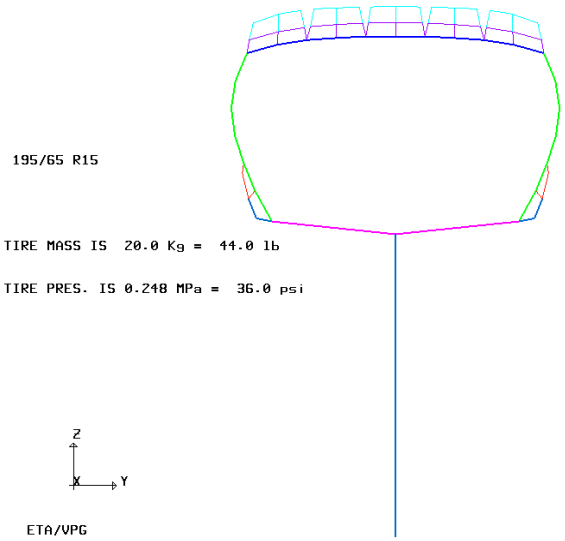
Description: This option allows the user to auto-generate composite tire models based on user-defined parameters.

Usage:

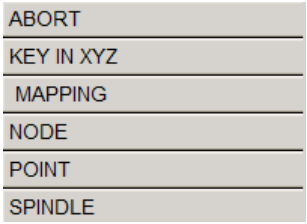
1. VPG displays a pop-up window with tire parameter data and prompts:
   > CREATE TIRE PROFILE

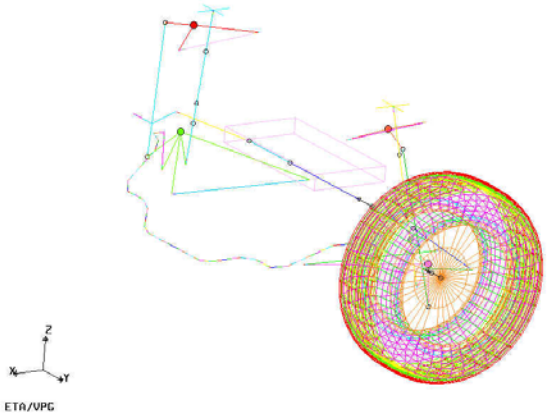


2. When the user has exited the tire parameter window, VPG displays the tire cross-section along with mass and pressure values and prompts:
   > PROCESSING ELEMENTS AND NODES
   > DO YOU ACCEPT THIS TIRE PROFILE. (Y/N/ABORT)
   - NO returns the user to the tire parameter window.

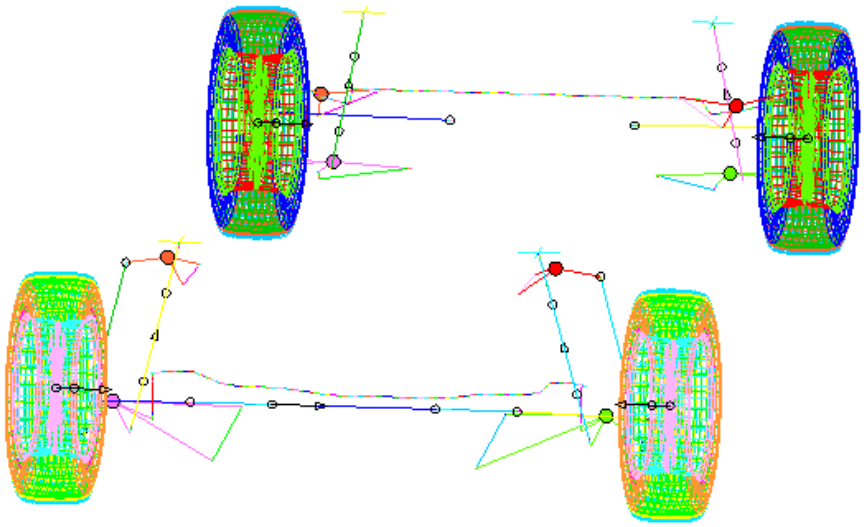


3. If the user accepts the tire profile, VPG prompts:
> DEFINE WHEEL CENTER LOCATION FOR TIRES





Note: For tire tuning procedures, see the eta/VPG applications manual, sections 1.5 and 2.4

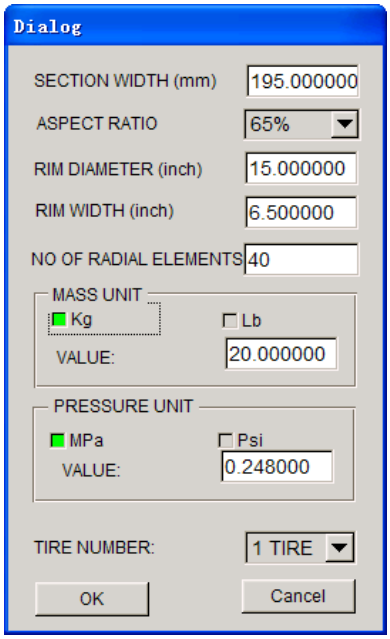


7.3.5 CREATE DIRECT MATRIX

Description: This option allows the user to auto-generate direct matrix tire models based on user-defined parameters.

Usage:

1. VPG displays a pop-up window with tire parameter data and prompts:
   > CREATE TIRE PROFILE

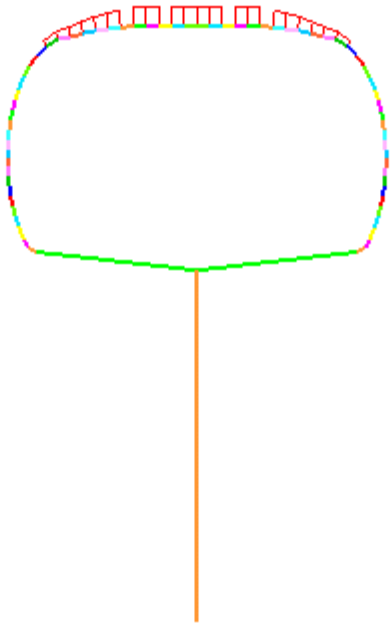


2. When the user has exited the tire parameter window, VPG displays the tire

cross-section along with mass and pressure values and prompts:

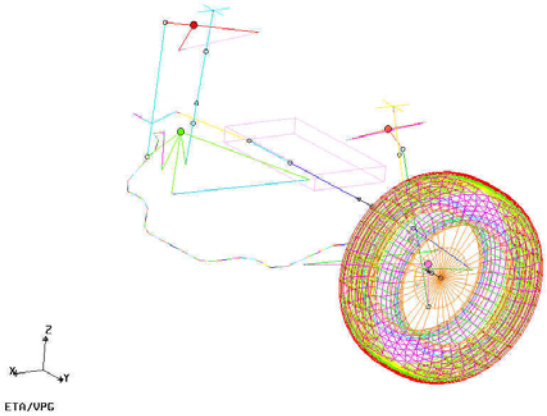
> **PROCESSING ELEMENTS AND NODES**
> **DO YOU ACCEPT THIS TIRE PROFILE. (Y/N/ABORT)**

- NO returns the user to the tire parameter window.

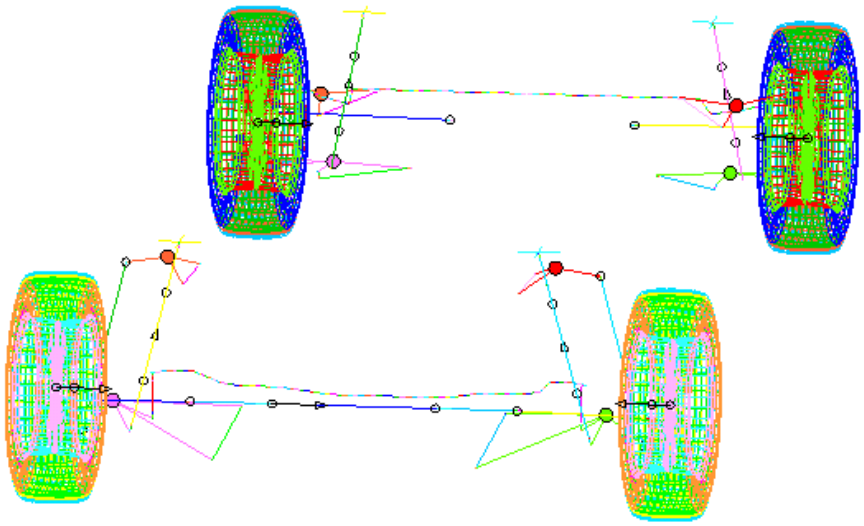


3. If the user accepts the tire profile, VPG prompts:
> DEFINE WHEEL CENTER LOCATION FOR TIRES

ABORT
KEY IN XYZ
MAPPING
NODE
POINT
SPINDLE



Note: For tire tuning procedures, see the eta/VPG applications manual, sections 1.5 and 2.4



7.4 Introduction to VPG/Safety

The automotive industry is legally required to design vehicles to meet the Federal Motor Vehicle Safety Standards (FMVSS) or ECE/EEC Homogenous Regulations and/or Corporate/Industry Standards. Various experimental and simulation approaches have been adopted to achieve these requirements.

The most commonly performed vehicle crashworthiness and occupant safety simulations are somewhat "standardized" and various groups using different pre-/post-processors have established what are known as "best practice" procedures.

ETA's consulting group has been performing simulations on vehicle development programs for over a decade and developed the crash tools and simulation technology to meet customer requirements throughout the process.

The VPG/Safety Module is organized into three (3) components:

1. **Crash Tool Library**: consists of all necessary crash tool models, dummy models, barrier models, impactors, etc.
2. **Vehicle Crashworthiness Processes**: includes FMVSS, ECE/EEC, and Industry Standards.
3. **Occupant Safety Processes**: includes FMVSS, ECE/EEC, and Industry Standards.

Details of these components and their capabilities are documented in the following sections.

Notice to Users:

While the following sections of this manual have been created using the publicly available information for each of the tests and test simulations described in this manual, the user should exercise caution when performing these analyses.

Current testing methods and governmental regulations should be studied and compared to the testing and analysis methods described here so that a fully accurate study can be performed.

The writers of this document and the authors of the VPG/Safety software strongly recommend that all users take the initiative to study the testing procedure and the related analysis procedures and take any necessary measures to assure the accuracy of the simulations created using VPG/Safety.

7.4.1 General Processing Procedure

The crash/safety simulations documented herein have similar requirements and setup procedures. They generally require a structure assembly, (either a full vehicle or a component of the vehicle), and at least one crash tool model (dummy, barrier, rigid wall etc.). During the simulation, either a structure moves at a particular speed into a crash tool or a crash tool impacts the structure. The displacement of the structure, the acceleration (or deceleration) pulse, the G-forces on the dummy models, and certain specified results are then post-processed and compared with the performance criteria and legislated requirements.

Therefore, the "canned simulations" have a general processing procedure. From case to case, the user will see variations in the components and setup procedure. However, the general process remains the same.

The user first establishes a validated structure model that includes proper mesh, material properties, contact definitions, and correct connectivity to satisfy the given modeling guidelines.

The structure model is then placed in simulation position; most model positions are equivalent to the vehicle design coordinate system. However, some are defined in an alternative "testing fixture" coordinate system.

The crash tool model is then placed in a particular location and in a certain direction. In most cases, the crash tool model position is given. Occasionally, users will have to define the location and direction if there is no default standard. In occupant safety cases, the dummy models are placed on the seat at the H-point or a test fixture position.

The prescribed impact velocity and direction are given as default values according to legislated criteria. The user is able to alter these values according to individual case test requirements.

The simulation results have to be output according to their case requirements. Therefore, the nodes, contact surfaces, and sections are defined either automatically or by user-defined node set, segment set, and section during the setup process. The output request and the frequency (sampling rate) are also defined for post-processing purposes.

Legislated criteria, such as HIC, TTI, and Femur Load are calculated in addition to the crash pulse, impact force, and contact force and can then be post-processed.

All simulations have more than one possible setup process. The VPG/Safety module offers the user great flexibility in the various utility routines to perform the simulations. This manual documents one recommended processing approach for each case to demonstrate the program's capability and to allow less experienced users to adopt a consistent methodology.

Each “Usage” contains a documented, proven procedure for setting-up the case, submitting the analysis, and post-processing the results. The approach is technically sound and will provide correct simulation results. Therefore, it is highly recommended that new users run their simulations according to this step-by-step manual at least once before developing their own procedure using the various utility features and functions.

7.4.2 Crash Tool Library

ETA has developed and maintained a set of crash tool models for the cases presented in this manual for consulting projects over the last decade. This set of crash tool models is included as part of the software license agreement and should not be used with other pre-/post-processors.

The crash module allows users the flexibility to store their own crash tool models in a customized

library for easy access or to save them in ASCII format for private use.

ETA can demonstrate First Technology Safety System (FTSS) dummy models for use with the module including Hybrid III 50% Male, 95% Male, 5% Female, and various Child Dummy models. However, these dummy models are distributed by FTSS and are not part of the eta/VPG license agreement.

ETA offers a set of "generic" FEA (Flexible Body and/or Rigid Body) dummy models that are tuned to test simulation results. Although the material properties, joint rates, and construction are not fully validated, they are still complete models developed and used in crash simulation studies.

ETA fully supports and maintains the FTSS (if properly licensed) and "generic" dummy models as part of the VPG/Safety licensing agreement.

A set of barrier models is included. These are also "generic" models developed by ETA and tuned to test results.

The Crash Tool Library is in ASCII format so that the LS-DYNA input file of each model is stored and can be modified and updated in the library per user criteria.

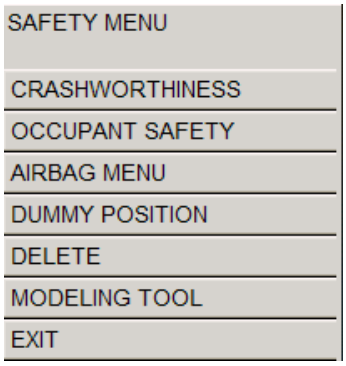
The user can also display models in the library to see a general description of the models and examine their characteristics using the VPG/Safety module. The crash tool models' material properties, contact definitions, node sets, segment sets, section sets, and other default definitions are automatically loaded as part of the input file and monitored throughout the crash safety/simulation procedure.

Pictures and a brief description of the crash tool models included in the VPG/Safety modules are documented in the following sections and are also included in the Step-by-Step simulation procedure when applicable.

The details, development process, and validation data are available upon request.

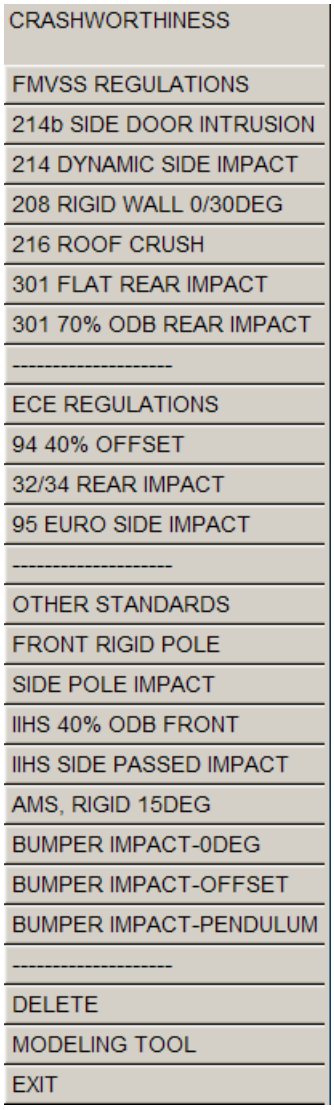
7.4.3 VPG/Safety Menu

The following functions are included in the VPG/SAFETY menu:



A detailed description of each function is given in the following sections.

7.4.4 Crashworthiness Menu


CRASHWORTHINESS
FMVSS REGULATIONS
214b SIDE DOOR INTRUSION
214 DYNAMIC SIDE IMPACT
208 RIGID WALL 0/30DEG
216 ROOF CRUSH
301 FLAT REAR IMPACT
301 70% ODB REAR IMPACT

ECE REGULATIONS
94 40% OFFSET
32/34 REAR IMPACT
95 EURO SIDE IMPACT

OTHER STANDARDS
FRONT RIGID POLE
SIDE POLE IMPACT
IIHS 40% ODB FRONT
IIHS SIDE PASSED IMPACT
AMS, RIGID 15DEG
BUMPER IMPACT-0DEG
BUMPER IMPACT-OFFSET
BUMPER IMPACT-PENDULUM

DELETE
MODELING TOOL
EXIT

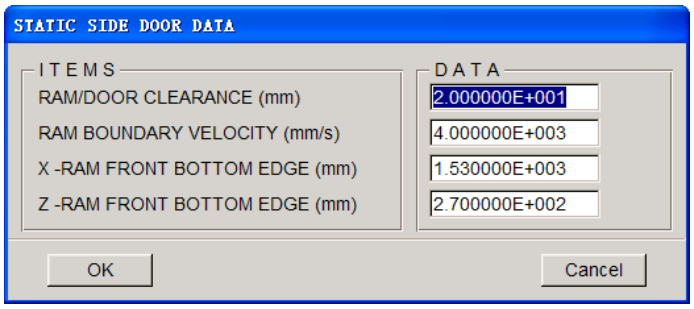
7.4.5 FMVSS REGULATIONS

7.4.5.1 FMVSS214b SIDE DOOR INTRUSION

Description: This function allows the user to generate an FMVSS214b Static Side Door Intrusion impact ram and position it within the current VPG database.

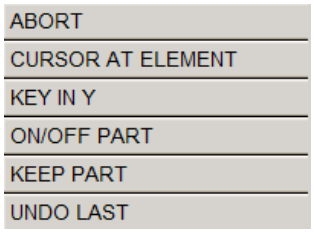
Usage:

1. VPG prompts:
   > ENTER STATIC SIDE DOOR INTRUSION INFORMATION



   Input values and press OK.

2. VPG prompts:
   > PICK THE FRONT EDGE IN THE LOWER SURFACE OF THE RAM
   > PICK A NODE OR INPUT THE Y VALUE OF THE RAM



3. Select CURSOR AT ELEMENT (default), and select an element from the door outer panel as target position. The rigid ram will be positioned to desired location. Proper contact and velocity will also be defined in the model.

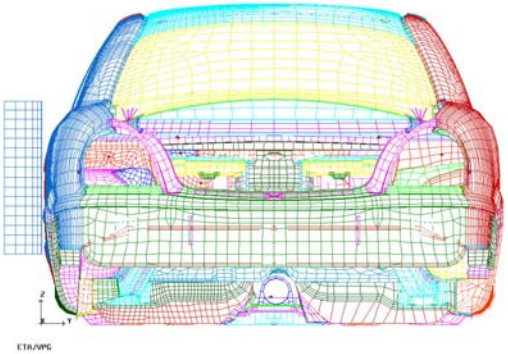


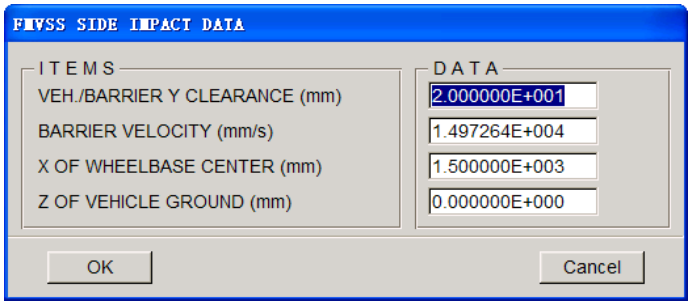
Figure 1 An Example of a Generated Side Door Intrusion Model

7.4.5.2 FMVSS214 DYNAMIC SIDE IMPACT

Description: This function allows the user to import the FMVSS214 US Moving Deformable Barrier (MDB) and dummies, and position these according to the requirements of FMVSS 214.

Usage:

1. VPG prompts:
   > ENTER FMVSS214 SIDE IMPACT INFORMATION



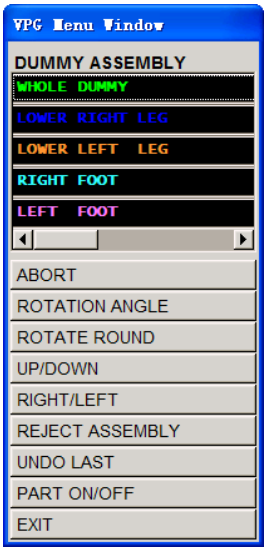
2. Input values and press OK.

3. VPG prompts:
   > PICK A NODE TO CALCULATE BARRIER/VEHICLE CLEARANCE
   - Select a node from the left-most part of the vehicle.
   - The MDB is imported from the library and put on the specified location.

4. VPG prompts for the location of the dummy.
   > PICK TEST POINT/H-POINT FOR DRIVER FMVSS214 DUMMY.
   - The H-POINT is a reference point of the dummy based on which the dummy is located.

5. Select KEY IN X, Y, Z and key in input values. The US SIDE IMPACT DUMMY is imported from the library and put on the specified location.

6. A ROTATION menu appears in the VPG Menu Window. Using these menu options, the user may perform 2-D positioning of the dummy, rotating the dummy body, tibia (lower leg) or foot position.



7. Select EXIT.

8. VPG prompts the location of passenger dummy:
   > PICK TEST POINT/H-POINT FOR PASSENGER FMVSS214 DUMMY.

9. Select KEY IN X, Y, Z and input values. The passenger dummy is imported and positioned according to the H-point.

Similar to the driver dummy, a rotation menu appears in the VPG Menu Window. The user can rotate the whole dummy or just the leg or foot to adjust their positions. Select EXIT to complete the dummy positioning.

The MDB and dummies will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

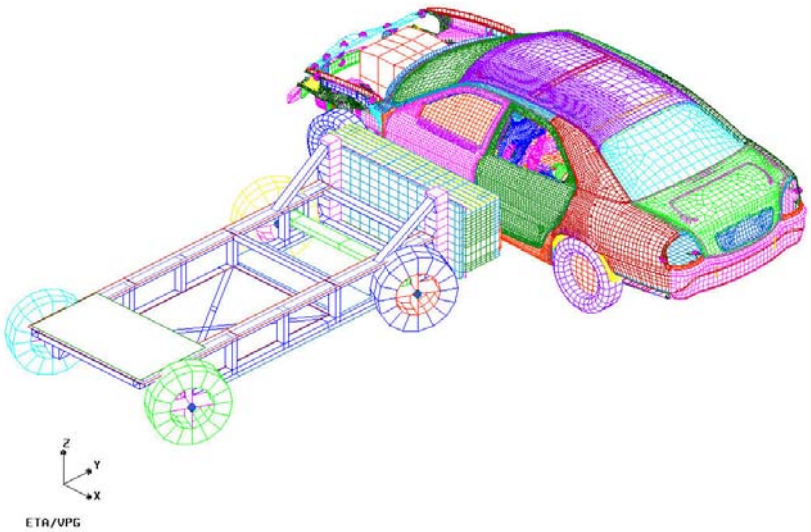


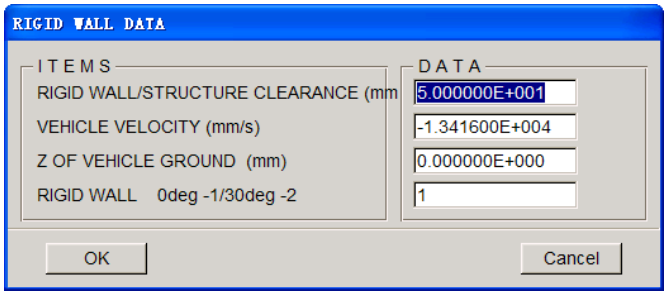
Figure 2 An Example of a Generated Dynamic Side Impact Model

7.4.5.3 FMVSS208 RIGID WALL 0/30 DEG

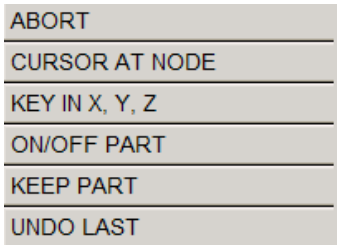
Description: This function allows users to create rigid walls to simulate the FMVSS208 0 Degree or 30 Degree Frontal Impact.

Usage:

1. VPG prompts:
   > **ENTER RIGID WALL CRASH INFORMATION**



2. VPG prompts:
   > **PICK VEHICLE FRONT BUMPER TO LOCATE RIGID WALL**



3. Select CURSOR AT ELEMENT (default), and select an element from the bumper part as the target position. The Frontal rigid wall and rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

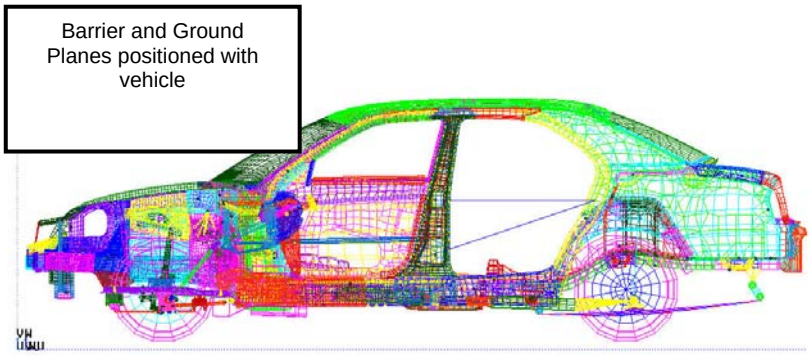


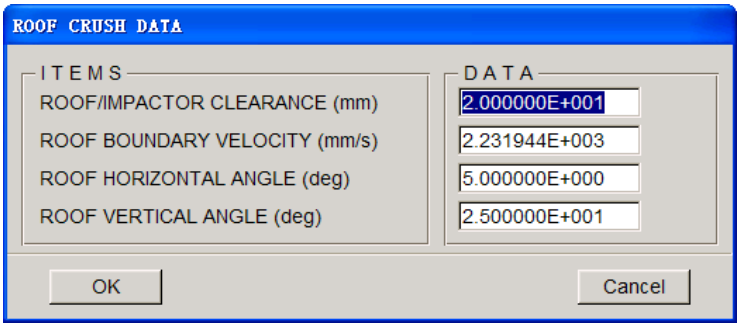
Figure 3 An Example of a Generated FMVSS208 Frontal Impact Model

7.4.5.4 FMVSS 216 ROOF CRUSH

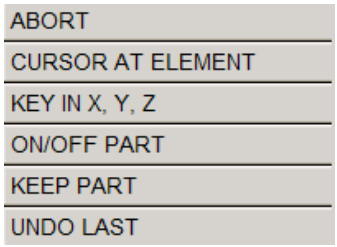
Description: This function allows users to create and position a rigid plate used to simulate the FMVSS216 roof crush test.

Usage:

1. VPG prompts:
   > **ENTER ROOF CRASH INFORMATION**



2. VPG prompts:
   > **PICK A CRASH TARGET CENTER TO LOCATE ROOF**



3. Select CURSOR AT ELEMENT (default), and select an element from the roof rail part as target position. The rigid plate is imported and placed in the location you have specified. Proper contacts and velocity will also be defined in the model.

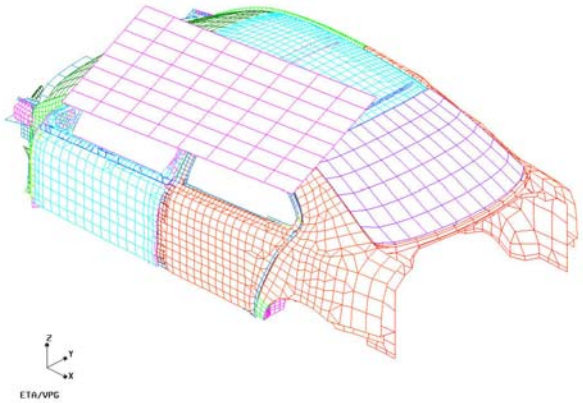


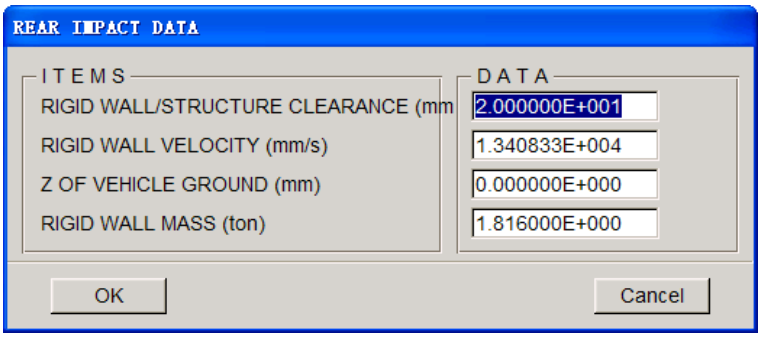
Figure 4 An Example of a Generated FMVSS216 Roof Crush Model

7.4.5.5 FMVSS301 FLAT REAR IMPACT

Description: This function imports and positions rigid walls used to simulate an FMVSS301 Rear Impact test.

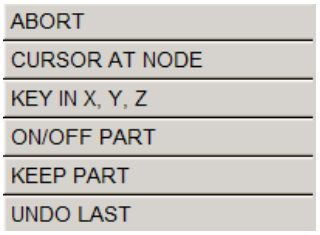
Usage:

1. VPG prompts:
   > ENTER REAR IMPACT INFORMATION



Input values and press OK.

2. VPG prompts:
   >PICK VEHICLE REAR BUMPER TO LOCATE RIGID WALL



3. Select CURSOR AT ELEMENT (default), and select an element from the part as the target position. The rear rigid wall and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

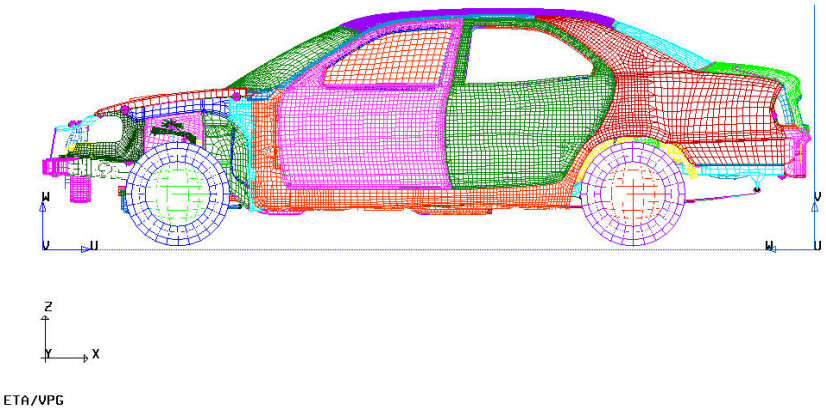


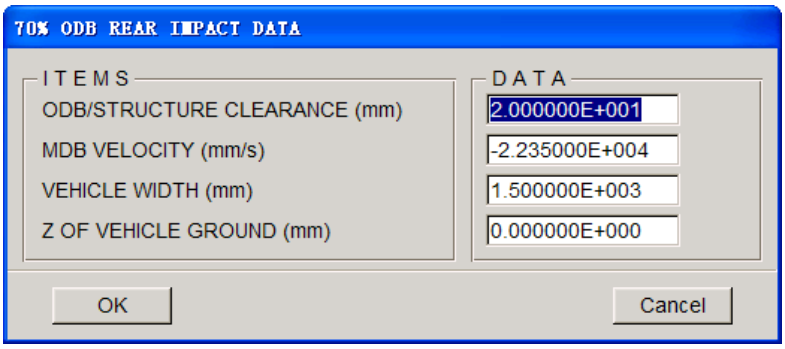
Figure 5 An Example of a Generated FMVSS301 Flat Rear Impact Model

7.4.5.6 FMVSS301 70% REAR IMPACT

Description: This function will generate a Moving Deformable Barrier and a rigid road surface to simulate FMVSS301 70% rear impact.

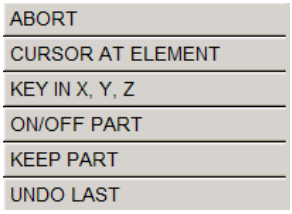
Usage:

1. VPG prompts:
   > ENTER REAR MDB INFORMATION



Input values and press OK.

2. VPG prompts:
   > PICK A TARGET CENTER TO LOCATE REAR MDB



.

3. Select CURSOR AT ELEMENT (default), and select an element from the rear bumper part as target position. The MDB and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

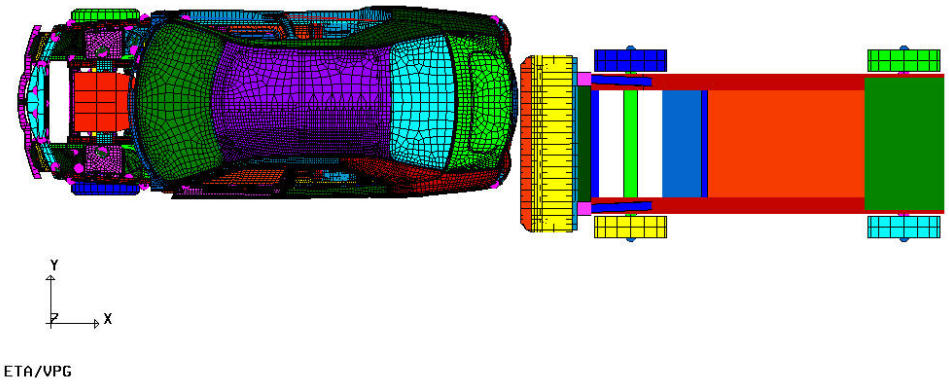


Figure 6 An Example of a Generated FMVSS301 70% Rear MDB Model

7.4.6 ECE REGULATIONS

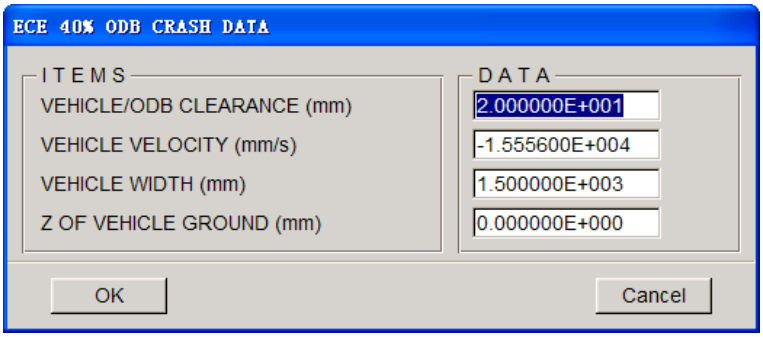
7.4.6.1 ECE 94 40% OFFSET FRONTAL IMPACT

Description: This function will create 40% ODB and a rigid road surface to simulate an ECE 94 Frontal Impact.

Usage:

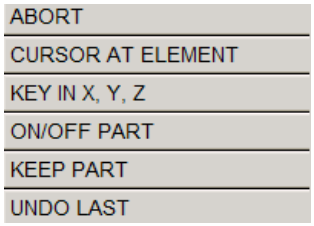
1. VPG prompts:

> ENTER ECE 40% ODB CRASH INFORMATION



Input values and press OK.

2. VPG prompts:
   > PICK VEHICLE FRONT BUMPER AGAINST ODB RIGHT-UP CORNER



3. Select CURSOR AT ELEMENT (default), and select an element from the front bumper part as target position. The ODB and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

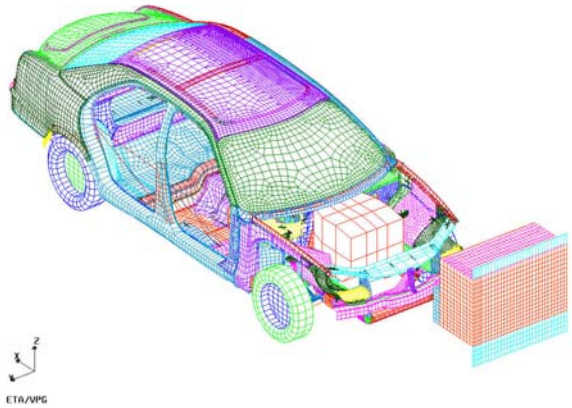


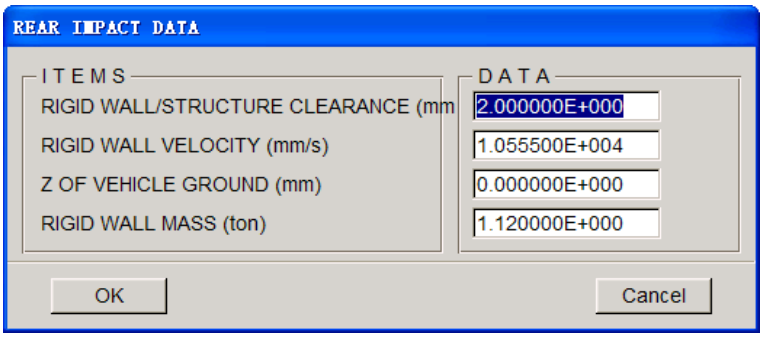
Figure 7 An Example of a Generated ECE 94 40% Front ODB Model

7.4.6.2 ECE 33/34 REAR IMPACT

Description: This function will create and position rigid walls that are used to simulate an ECE 33/34 Rear Impact test.

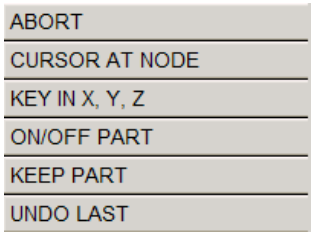
Usage:

1. VPG prompts:
   > ENTER REAR IMPACT INFORMATION



Input values and press OK.

2. VPG prompts:
   > PICK VEHICLE REAR BUMPER TO LOCATE RIGID WALL



3. Select CURSOR AT ELEMENT (default), and select an element from the part as target position. The rear rigid wall and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

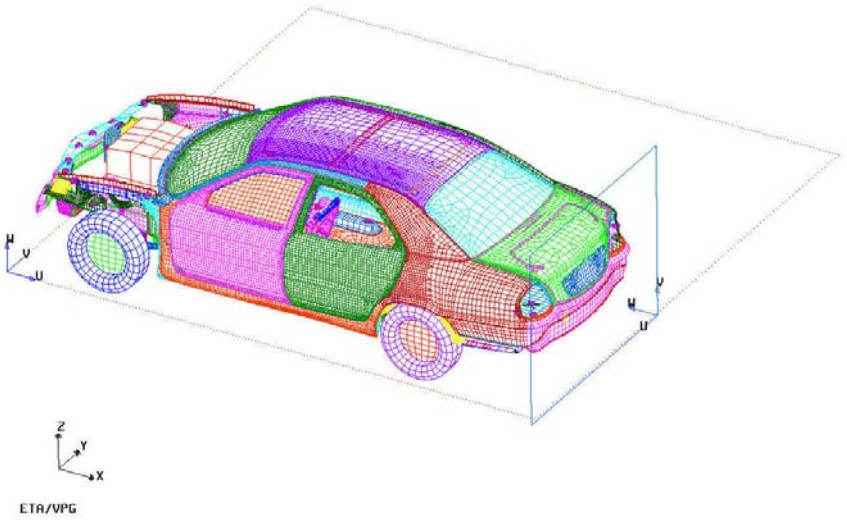


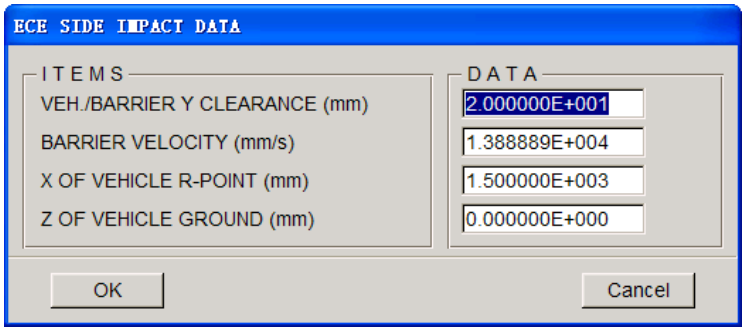
Figure 8 An Example of a Generated ECE33/34 Rear Impact Model

7.4.6.3 ECE 95 EURO SIDE IMPACT

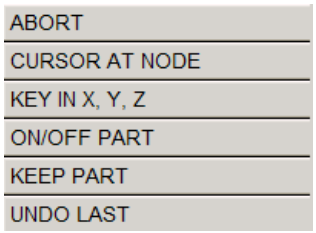
Description: This function will generate a Moving Deformable Barrier and a rigid road surface to simulate an ECE95 Side Impact.

Usage:

1. VPG prompts:
   > ENTER ECE95 SIDE IMPACT INFORMATION



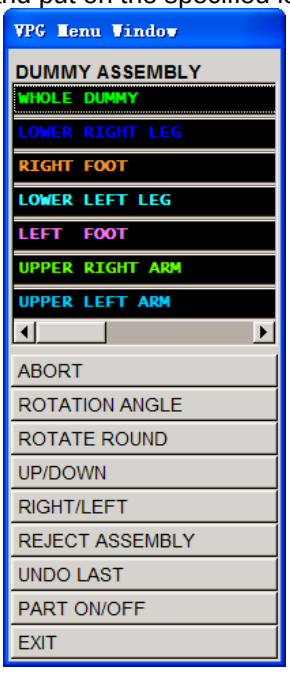
2. VPG prompts for the crash target for MDB:
   > PICK A NODE TO CALCULATE BARRIER/VEHICLE CLEARANCE



3. Select CURSOR AT ELEMENT (default), and select an element from the part as target position. The MDB and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

4. VPG prompts for the location of the driver dummy:
   >PICK TEST POINT/H-POINT FOR ECE95 DUMMY

5. Select KEY IN X, Y, Z, and input values. The EUROPEAN SIDE IMPACT DUMMY is imported from the library and put on the specified location.



6. A ROTATION menu appears in the VPG Menu Window. The user can rotate the dummy or leg to adjust their position. The default rotation angle increment is 1 degree. You can select CHANGE ANGLE to change it.

7. Select EXIT to finish Dummy Position.

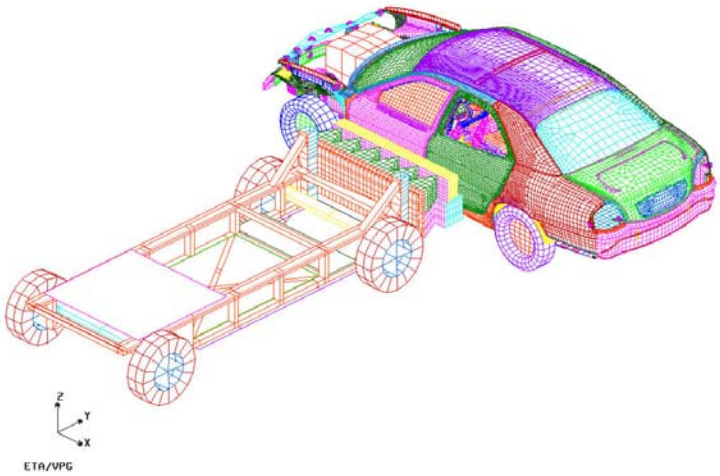


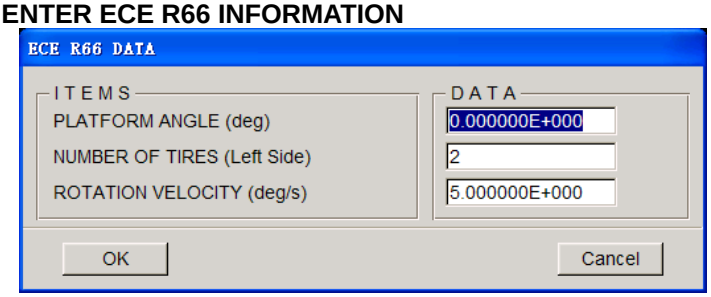
Figure 9 An Example of a Generated ECE95 Dynamic Side Impact Model

7.4.6.4 ECE R66

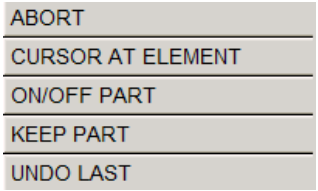
Description: This function will generate ground, platform and supporters to simulate an ECE R66 bus roll over.

Usage:

1. VPG prompts:
   > ENTER ECE R66 INFORMATION



2. VPG prompts to locate the supporters:
   > PICK FRONT LEFT TIRE TO LOCATE FIRST SUPPORTER



3. Select CURSOR AT ELEMENT (default), and select an element from front left tire, VPG prompts:
   > PICK REAR LEFT TIRE TO LOCATE SECOND SUPPORTER

4. Select an element from rear left tire, the supporters, ground and a platform will be

positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

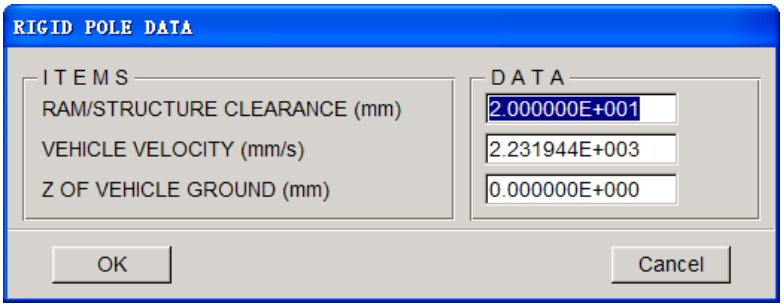
7.4.7 OTHER STANDARDS

7.4.7.1 FRONT RIGID POLE

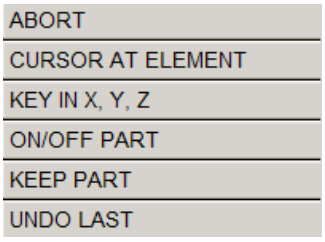
Description: This function will create a rigid pole at the front of the vehicle.

Usage:

1. VPG prompts:
   > ENTER RIGID POLE CRASH INFORMATION

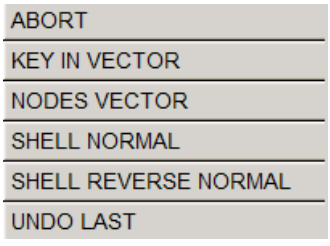


2. VPG prompts:
   > PICK A CRASH TARGET CENTER TO LOCATE FRONT POLE



   Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:
   > DEFINE CONTACT NORMAIL DIRECTION AGAINST POLE



4. VPG prompts:
   > ALL NODES ARE SLAVE TO RIGID WALL
   > INFINITE SIZE PLANE IS DEFINED

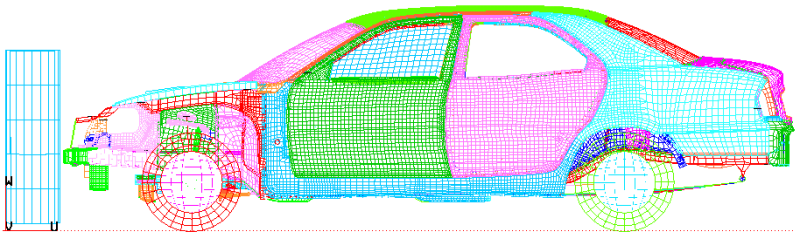


Figure 10 An Example of a Generated Front Rigid Pole Impact Model

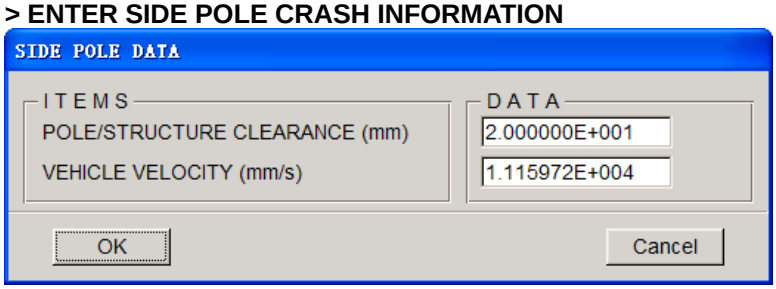
7.4.7.2 SIDE POLE IMPACT

Description: This function will create a rigid pole at the side of the vehicle.

Usage:

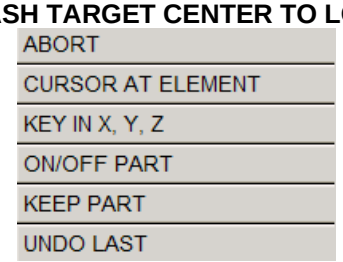
1. VPG prompts:

   > ENTER SIDE POLE CRASH INFORMATION



2. VPG prompts:

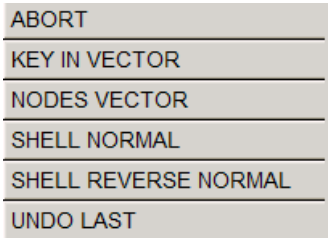
   > PICK A CRASH TARGET CENTER TO LOCATE SIDE POLE



   Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:

   > DEFINE CONTACT NORMAL DIRECTION AGAINST POLE



   There are four approaches to define the head direction:

   - KEY IN VECTOR Manually key in a vector.

• NODES VECTOR	Select two nodes to define a vector.
• SHELL NORMAL	The head will align with the normal of the element selected.
• SHELL REVERSE NORMAL	The head will align with the reverse normal direction of the element selected.

4. Select SHELL NORMAL.

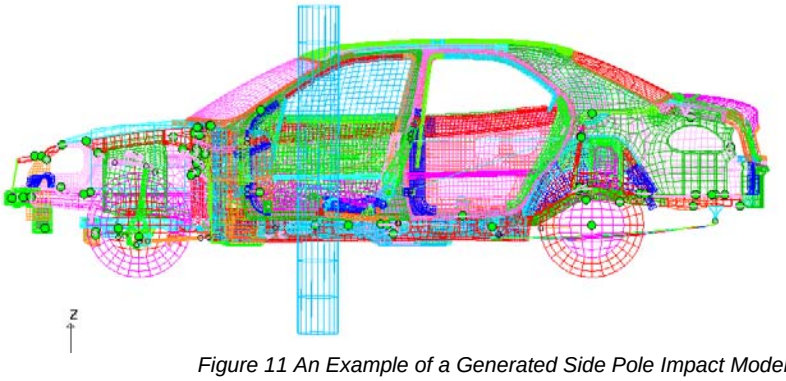


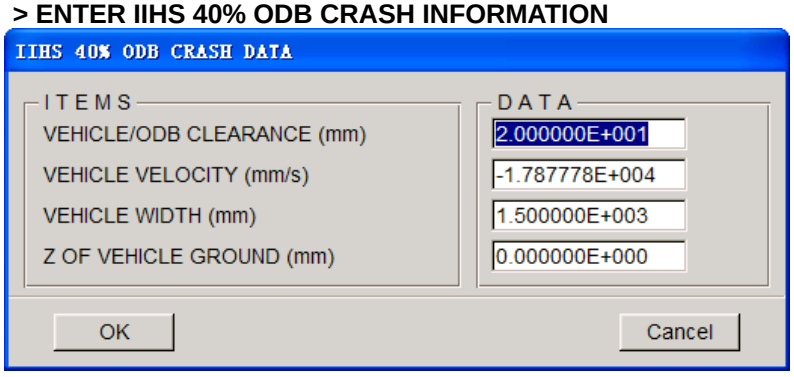
Figure 11 An Example of a Generated Side Pole Impact Model

7.4.7.3 IIHS 40% ODB FRONT

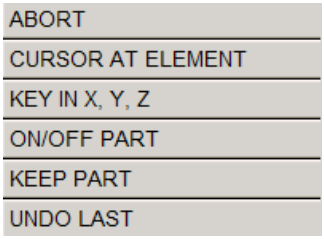
Description: This function will create ODB at the 40% front of the vehicle.

Usage:

1. VPG prompts:
   > ENTER IIHS 40% ODB CRASH INFORMATION



2. VPG prompts:
   > PICK VEHICLE FRONT BUMPER AGAINST ODB RIGHT-UP CORNER



   Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:
   > ALL NODES ARE SLAVE TO RIGID WALL
   > INFINITE SIZE PLANE IS DEFINED

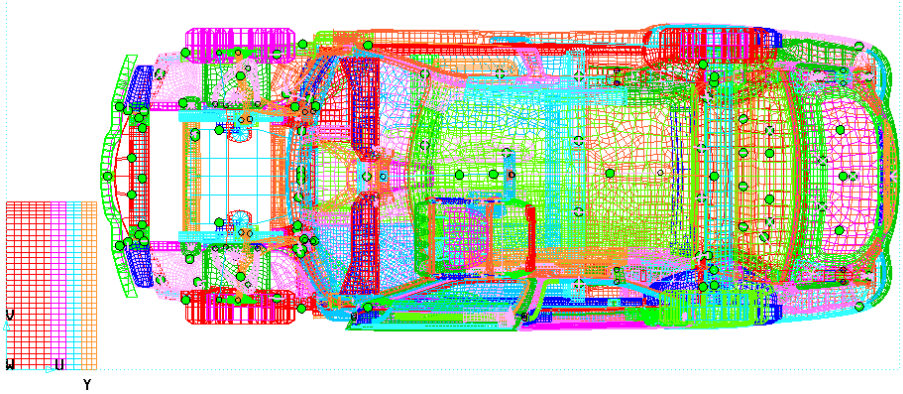


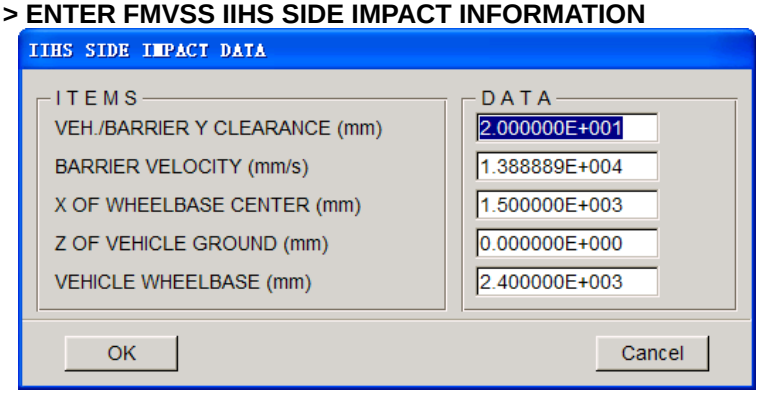
Figure 12 An Example of a Generated IIHS 40% ODB Impact Model

7.4.7.4 IIHS SIDE PASSED IMPACT

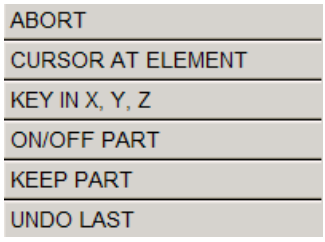
Description: This function will create ODB at the side of the vehicle.

Usage:

1. VPG prompts:
   > ENTER FMVSS IIHS SIDE IMPACT INFORMATION



2. VPG prompts for the crash target for MDB:
   > PICK A NODE TO CALCULATE BARRIER/VEHICLE CLEARANCE



3. Select CURSOR AT ELEMENT (default), and select an element from the part as target position. The MDB and a rigid road surface will be positioned in the desired locations. Proper contacts and velocities will also be defined in the model.

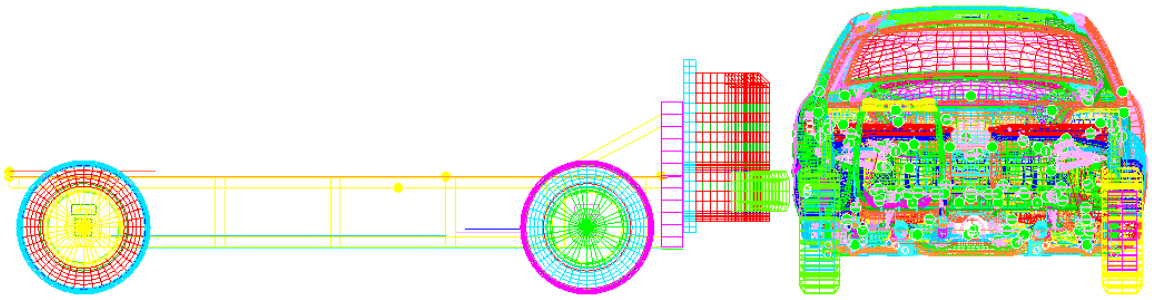


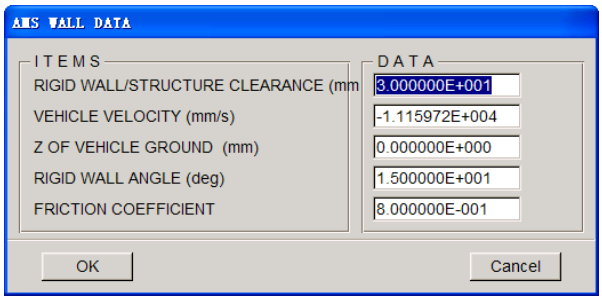
Figure 13 An Example of a Generated IIHS SIDE PASSED IMPACT Model

7.4.7.5 AMS, RIGID 15 DEG

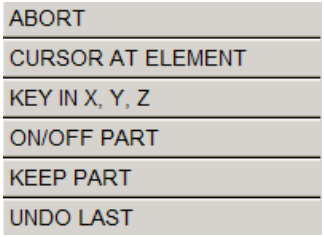
Description: This function will create a rigid wall at the front of the vehicle.

Usage:

1. VPG prompts:
   > ENTER AMS WALL INFORMATION



2. VPG prompts:
   > PICK VEHICLE FRONT BUMPER TO LOCATE RIGID WALL



   Select CURSOR AT ELEMENT (default), and select an element from the part as the target position.

3. VPG prompts:
   > ALL NODES ARE SLAVE TO RIGID WALL
   > INFINITE SIZE PLANE IS DEFINED

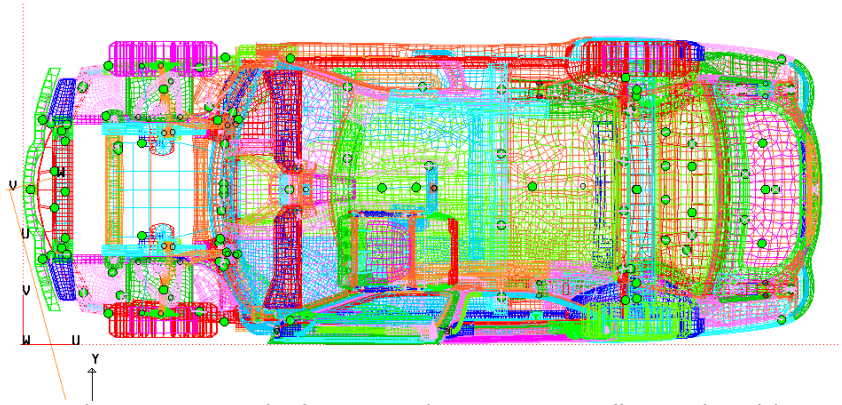


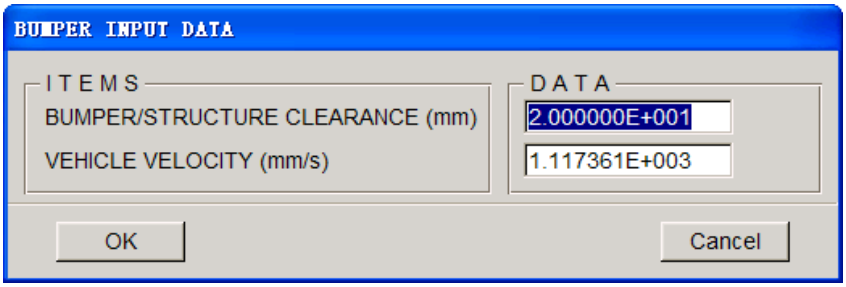
Figure 14 An Example of a Generated AMS 15 Degree Offset Crash Model

7.4.7.6 BUMPER IMPACT – 0 DEG

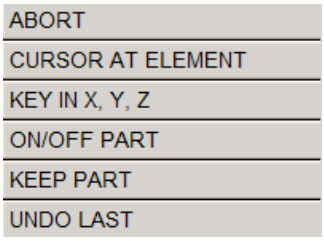
Description: This function will create a rigid bumper at the front of the vehicle.

Usage:

1. VPG prompts:
   > ENTER BUMPER CRASH INFORMATION

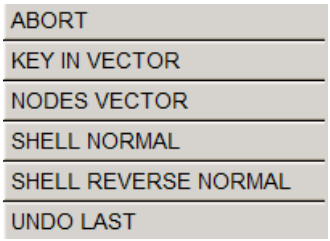


2. VPG prompts:
   > PICK A CRASH TARGET CENTER TO LOCATE BUMPER



   Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:
   > DEFINE CONTACT NORMAL DIRECTION AGAINST BUMPER



   There are four approaches to define the head direction:

• KEY IN VECTOR	Manually key in a vector.
• NODES VECTOR	Select two nodes to define a vector.
• SHELL NORMAL	The head will align with the normal of the element selected.
• SHELL REVERSE NORMAL	The head will align with the reverse normal direction of the element selected.

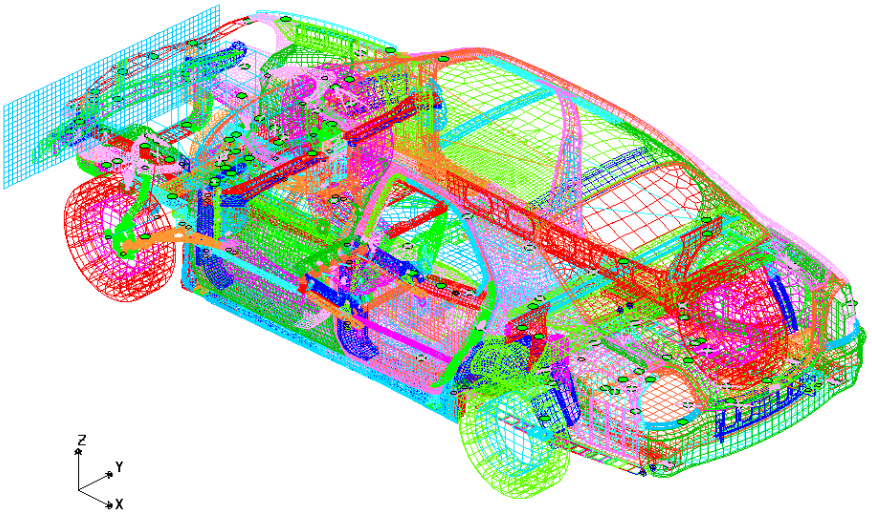


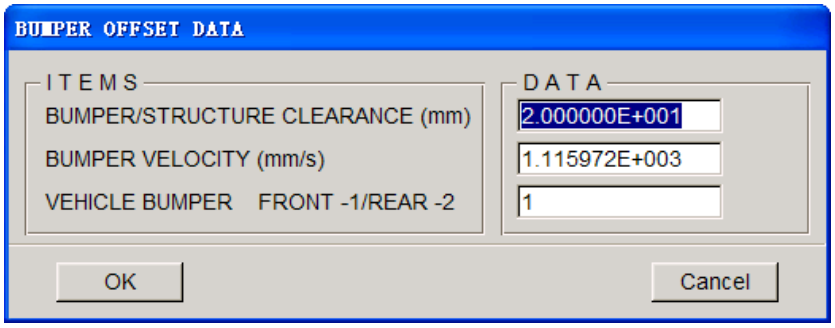
Figure 15 An Example of a Generated Rigid Wall Bumper Impact Model

7.4.7.7 BUMPER IMPACT-OFFSET

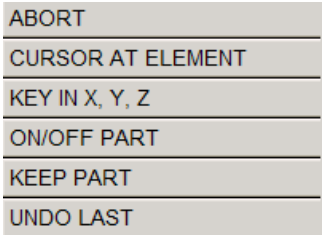
Description: This function will create a rigid pendulum in the front or rear of the vehicle.

Usage:

1. VPG prompts:
   > ENTER BUMPER CRASH INFORMATION



2. VPG prompts:
   > PICK A TARGET CENTER TO LOCATE PENDULUM



Select CURSOR AT ELEMENT (default), and select an element from the part as the target position.

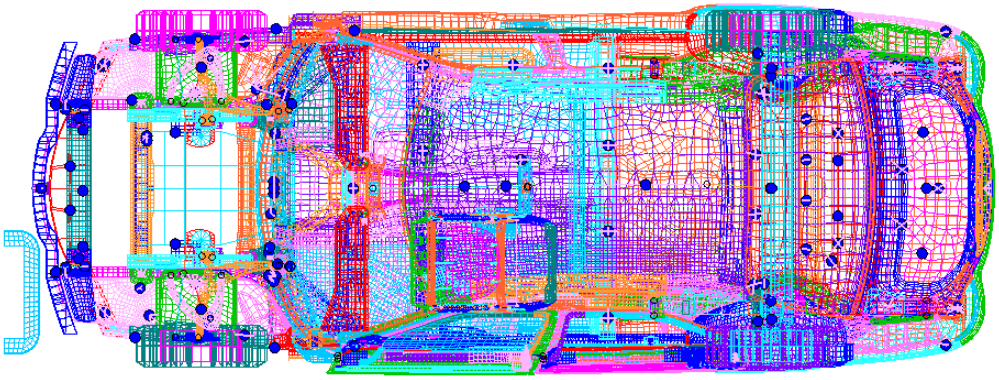


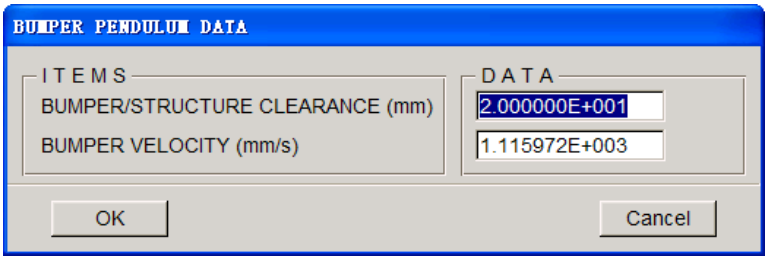
Figure 16 An Example of a Generated 0 Degree Pendulum Bumper Impact

7.4.7.8 BUMPER IMPACT-PENDULUM

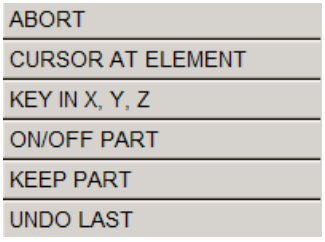
Description: This function will create a rigid pendulum in front of the vehicle.

Usage:

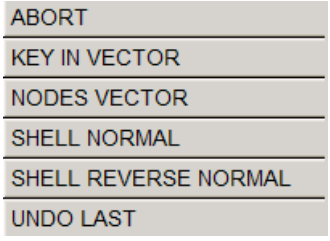
1. VPG prompts:
   > ENTER BUMPER CRASH INFORMATION



2. VPG prompts:
   > PICK A CRASH TARGET CENTER TO LOCATE BUMPER



3. VPG prompts:
   > DEFINE CONTACT NORMAL DIRECTION AGAINST BUMPER



   There are four approaches to define the normal direction:

• KEY IN VECTOR	Manually key in a vector.
• NODES VECTOR	Select two nodes to define a vector.
• SHELL NORMAL	The head will align with the normal of the element selected.
• SHELL REVERSE NORMAL	The head will align with the reverse normal direction of the element selected.

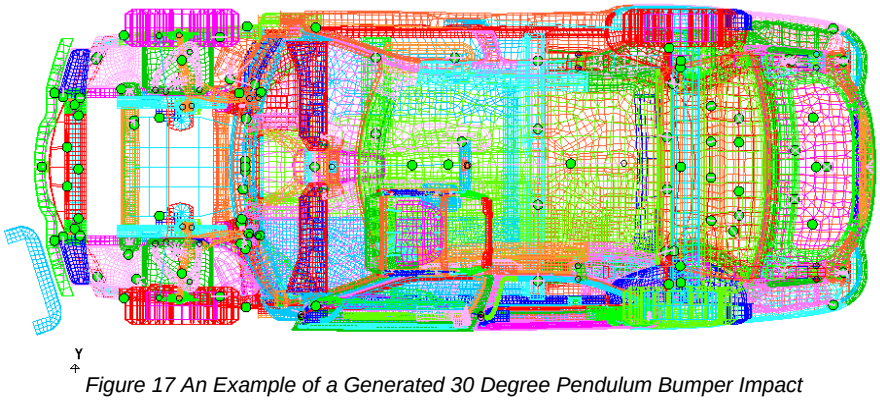


Figure 17 An Example of a Generated 30 Degree Pendulum Bumper Impact

7.5 OCCUPANT SAFETY

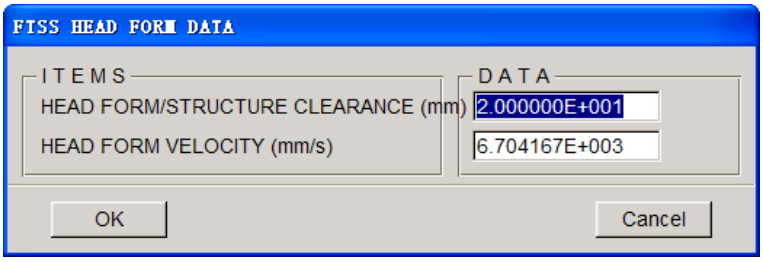
7.5.1 FMVSS REGULATIONS

7.5.1.1 FMVSS201 HEAD IMPACT

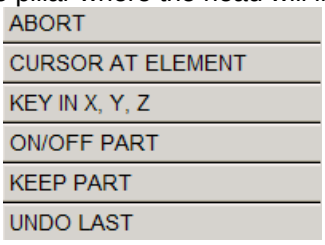
Description: This function will position a FMVSS201 Head Impact Head Form model in the VPG model database and apply appropriate boundary conditions. It should be noted that the FTSS Head Form is available only to users who have a valid FTSS Dummy or Head Form License Agreement.

Usage:

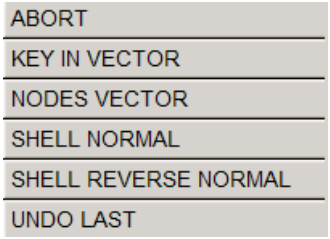
1. VPG prompts:
   > ENTER HEAD IMPACT INFORMATION



2. VPG prompts:
   > PICK MOST OUTSIDE SKIN TARGET CENTER TO LOCATE HEAD
   Define the target on the pillar where the head will impact.



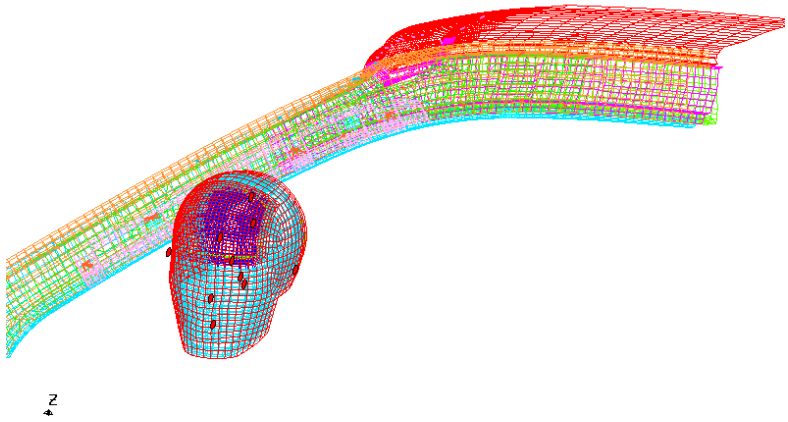
3. VPG prompts:
   > DEFINE CONTACT NORMAL DIRCETION AGAINST HEAD



   There are four approaches to define the head direction:

- KEY IN VECTOR — Manually key in a vector.
- NODES VECTOR — Select two nodes to define a vector.
- SHELL NORMAL — The head will align with the normal of the element selected.

- SHELL REVERSE NORMAL — The head will align with the reverse normal direction of the element selected.



7.5.1.2 201 ETA HEAD IMPACT

Description: This function will position a FMVSS201 Head Impact Head Form model in the VPG model database and apply appropriate boundary conditions. The FEA model used is an ETA head model. Please refer to FMVSS201 HEAD IMPACT for detail usage.

7.5.2.3 201 GM HEAD IMPACT

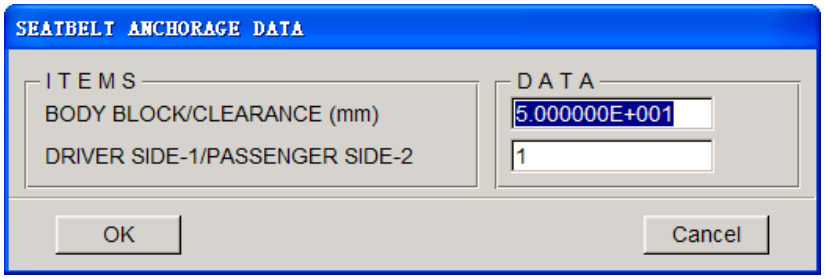
Description: This function will position a FMVSS201 Head Impact Head Form model in the VPG model database and apply appropriate boundary conditions. The FEA model used is a GM head model. Please refer to FMVSS201 HEAD IMPACT for detail usage.

7.5.1.4 FMVSS207/210 SEATBELT ANCHORAGE

Description: This function will import and position the body block used in the evaluation of seat belt anchorages to support the simulation of FMVSS 207 and FMVSS 210 requirements.

Usage:

1. VPG prompts:
   > ENTER SEATBELT ANCHORAGE CRASH INFORMATION



2. VPG prompts:
   > PICK A NODE TO LOCATE CENTER OF LOWER BLOCK

ABORT
CURSOR AT ELEMENT
KEY IN X, Y, Z
ON/OFF PART
KEEP PART
UNDO LAST

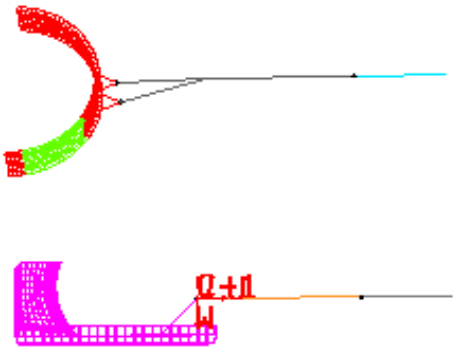
3. VPG prompts:
   > PICK A NODE TO LOCATE CENTER OF UPPER BLOCK

4. VPG prompts:
   > DEFINE A VECTOR TO MOVE SEATBELTS

ABORT
UPPER BLOCK
LOWER BLOCK
REVERSE OPERATION
KEY IN VECTOR
ON/OFF PART
PREVIOUS MENU
EXIT

5. VPG prompts:
   > DEFINE AN AXIS TO ROTATE SEATBELTS

ABORT
UPPER BLOCK
LOWER BLOCK
REVERSE OPERATION
ROTATION ANGLE
DEFINE AXIS
ON/OFF PART
PREVIOUS MENU
EXIT

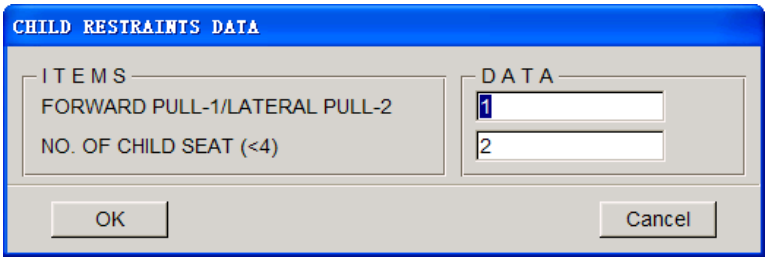


7.5.1.5 FMVSS225 CHILD RESTRAINTS ANCHORAGE

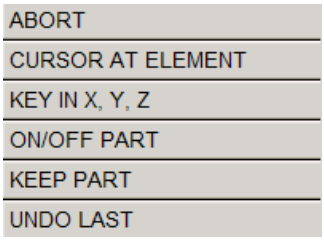
Description: This function will create child restraints anchorage in the model.

Usage:

1. VPG prompts:
   > ENTER CHILD RESTRAINTS INFORMATION

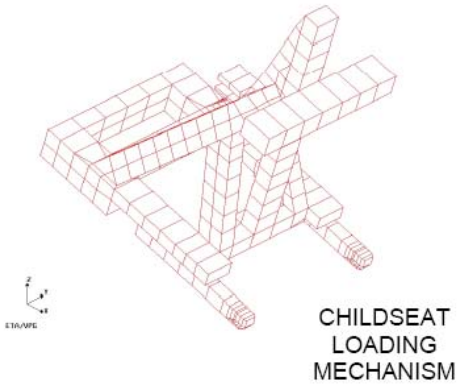


2. VPG prompts:
   > PICK A NODE TO LOCATE FIRST SEAT LEFT SIDE CLAMP



   Select CURSOR AT ELEMENT (default), and select an element from the part as the target position.

3. VPG prompts:
   > PICK A NODE TO LOCATE FIRST SEAT RIGHT SIDE CLAMP

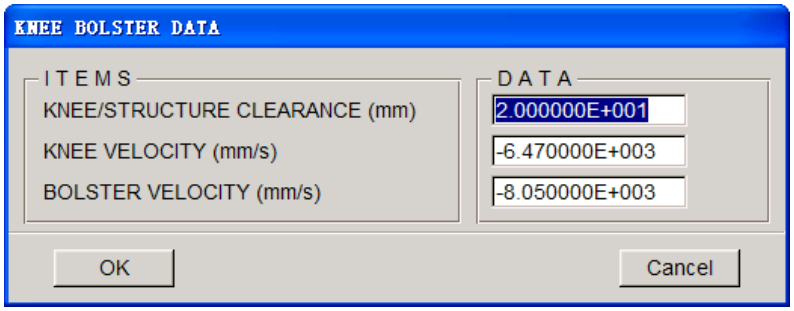


7.5.1.6 FMVSS208 KNEE BOLSTER

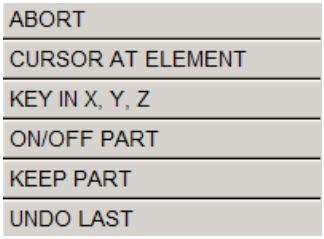
Description: This function will create knee impactors and position them in the model in order to conduct a knee bolster analysis.

Usage:

1. VPG prompts:
   > ENTER KNEE BOLSTER CRASH INFORMATION

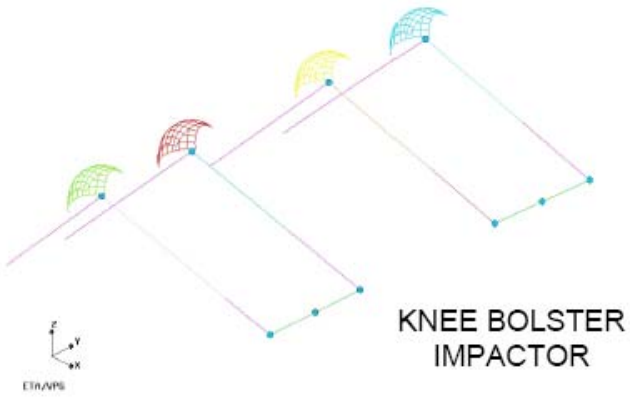


2. VPG prompts:
   > PICK A CRASH TARGET CENTER TO LOCATE DRIVER LEFT KNEE



   Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:
   > PICK A CRASH TARGET CENTER TO LOCATE PASSAGER LEFT KNEE

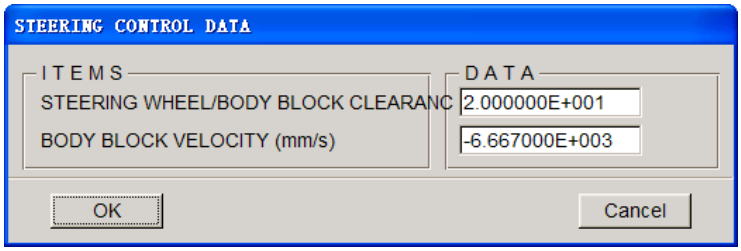


7.5.1.7 FMVSS203 STEERING CONTROL

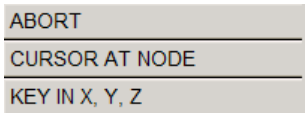
Description: This function will create a body block model and position it for simulation of an FMVSS 203 Steering Control System analysis.

Usage:

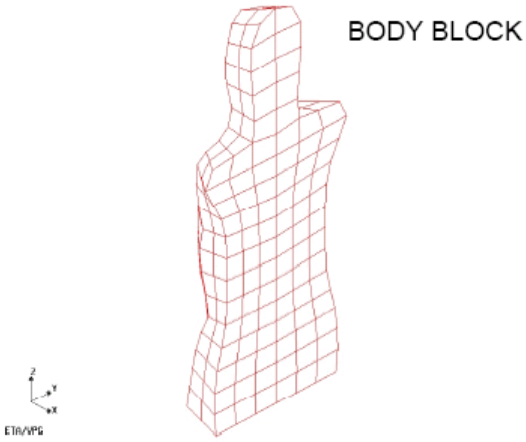
1. VPG prompts:
   >ENTER STEERING CONTROL INFORMATION



2. VPG prompts:
>PICK A STEERING WHEEL NODE TO LOCATE BODY BLOCK



Select CURSOR AT NODE (default), and select a node from the part as target position.



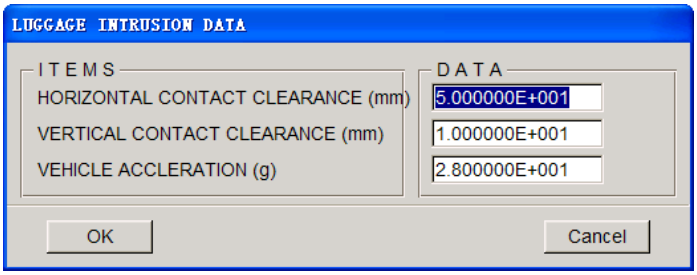
7.5.2 OCCUPANT SAFETY- ECE REGULATIONS

7.5.2.1 ECE17 LUGGAGE INTRUSTION

Description: This function will import and position luggage blocks of the proper weight and dimensions for use in an ECE 17 Luggage Intrusion simulation.

Usage:

1. VPG prompts:
> ENTER LUGGAGE INTRUSION INFORMATION



2. VPG prompts:

> PICK SEAT REAR-END TO LOCATE LUGGAGE HORIZONTALLY.

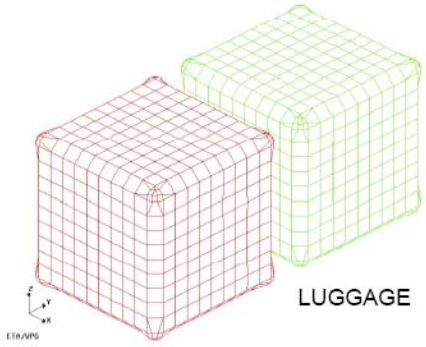
ABORT
CURSOR AT ELEMENT
KEY IN X, Y, Z
ON/OFF PART
KEEP PART
UNDO LAST

Select CURSOR AT ELEMENT (default), and select an element from the part as target position.

3. VPG prompts:

> PICK A TARGET CENTER TO LOCATE LUGGAGE VERTICALLY

ABORT
CURSOR AT ELEMENT
KEY IN X, Y, Z
ON/OFF PART
KEEP PART
UNDO LAST



7.6 DUMMY POSITIONING

The functions in the DUMMY POSITIONING menu allow the user to auto-generate dummy and seatbelt models from a library of dummy types and position them in the current database. The data required for positioning of the dummy is the H-Point of the occupant position.

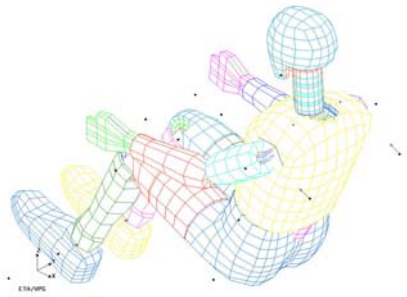
DUMMY MENU
VPG DUMMY
DEFORMABLE 50%
DEFORMABLE 5%
DEFORMABLE 95%
RIGID 50%
RIGID 5%
RIGID 95%
ES I
ES II
ES IIre
SID

OTHER DUMMY
TAKATA 3-YEAR CHILD
FAA HYBRID III
THOR LX DUMMY
LSTC 50% RIGID

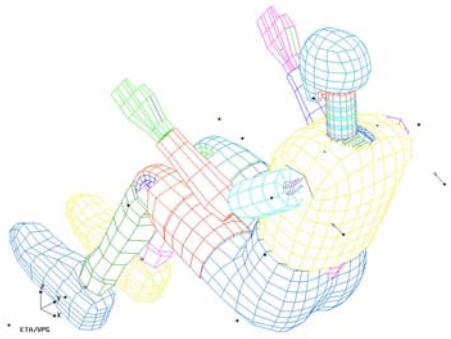
SEATBELT
EXIT

The user may select any dummy from the DUMMY MENU.

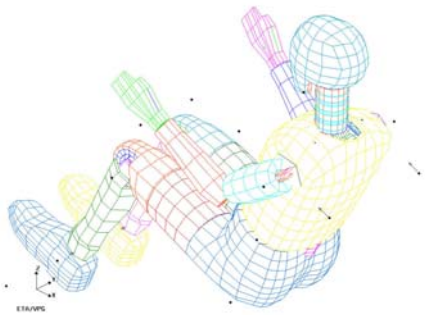
7.6.1 LSTC DUMMIES



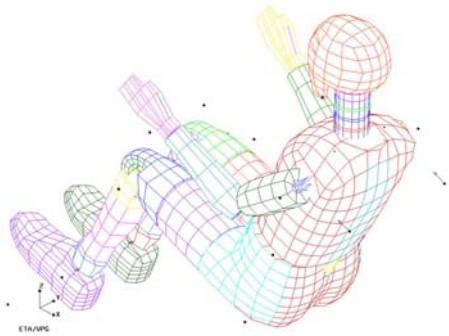
LSTC Deformable 50% Dummy Model



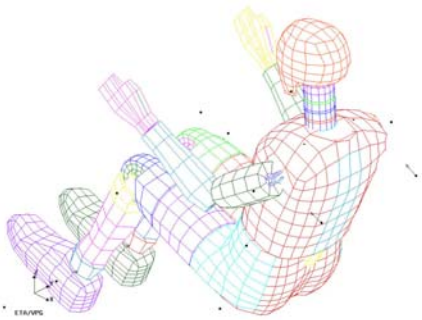
LSTC Deformable 95% Dummy Model



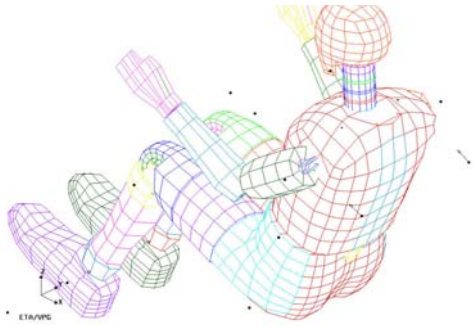
**LSTC Deformable 5%
Dummy Model**



**LSTC Rigid 5%
Dummy Model**

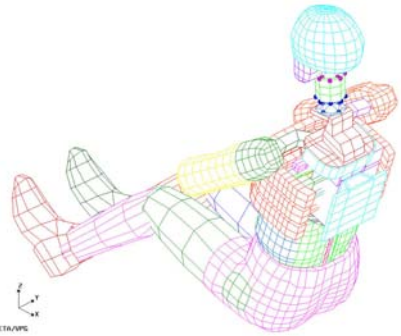


**LSTC Deformable 50%
Dummy Model**

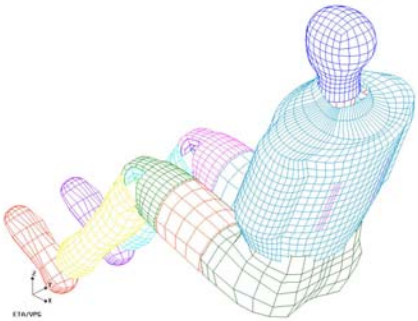


**LSTC Rigid 95%
Dummy Model**

7.6.2 ETA DUMMIES

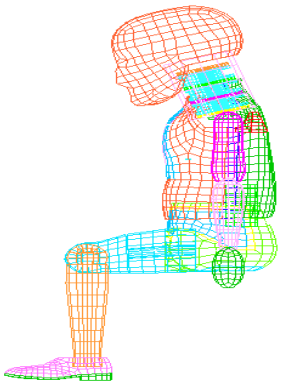


ETA ECE SIDE Model

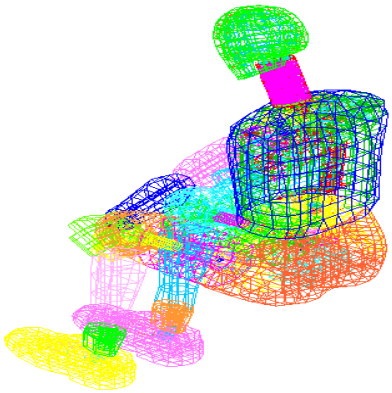


ETA US SIDE Model

7.6.3 OTHER DUMMIES



TAKATA 3 Year Child Model



FAA HYBRID III

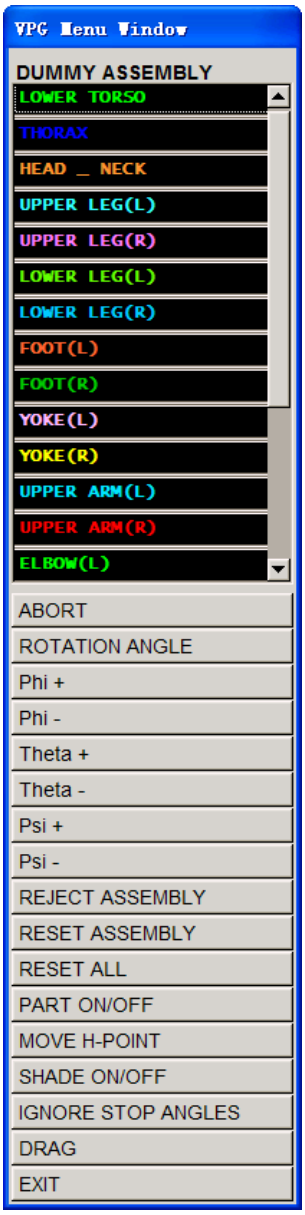
7.6.4 CREATE DUMMY MODELS

Description: This command imports and positions a finite element dummy model into the current model database.

Usage:

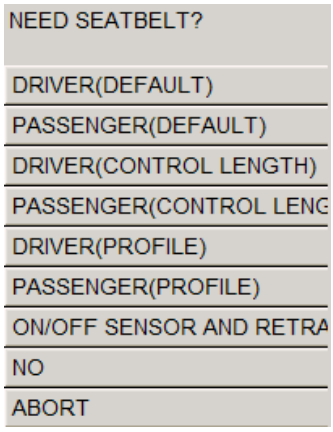
1. VPG prompts:
   > PICK TEST POINT/H-POINT FOR THE FTSS50 DUMMY

2. DUMMY ASSEMBY:



The user may adjust part altitude in the dummy, then press exit to end the adjustment procedure.

3. VPG prompts:
> DO YOU NEED A SEATBELT?



The user can choose a seatbelt type in this menu.

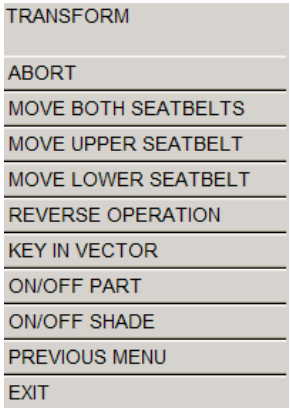
4. VPG prompts:
>PICK A NODE ON OUTBOARD UPPER ANCHOR

5. VPG prompts:
>PICK A NODE ON INBOARD LOWER ANCHOR

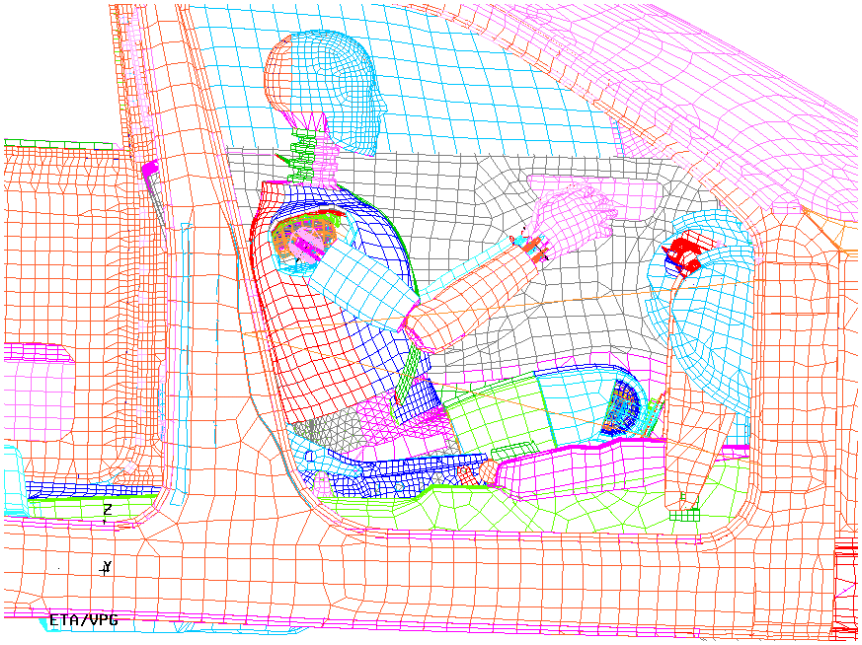
6. VPG prompts:
>PICK A NODE ON OUTBOARD LOWER ANCHOR

7. VPG prompts:
>PICK A NODE ON OUTBOARD LAP BELT ANCHOR

8. VPG prompts:
>DEFINE A VECTOR TO MOVE SEATBELTS



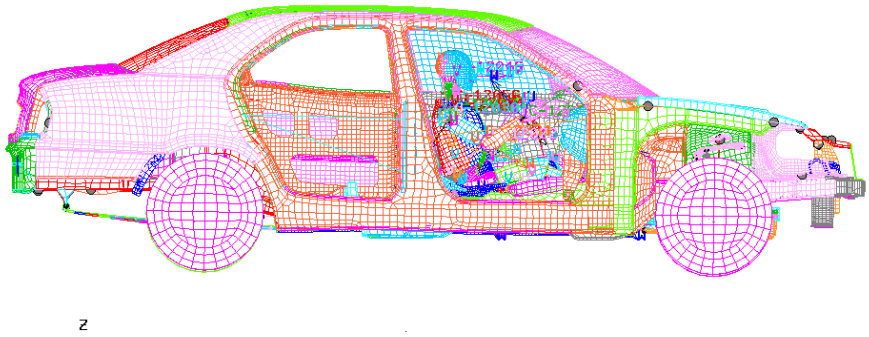
Use KEY IN VECTOR to define a vector, then move the upper or lower seatbelt to the right place.



9. VPG prompts:

>PICK A SEATBELT ELEMENT TO START PULL/PUSH

PULL/PUSH
ABORT
CURSOR AT ELEMENT
KEY IN AMOUNT
ON/OFF PART
PULL SEATBELT
PUSH SEATBELT
ON/OFF SHADE
PREVIOUS MENU
EXIT



7.7 DELETE MODEL

Description: This option allows the user to delete the currently defined crash models. Any tool available in the Crash Tool Library may be deleted. In the case where a dummy model and barrier model have been created using the same process, the DELETE function will delete both the dummy model and the barrier.

Usage:

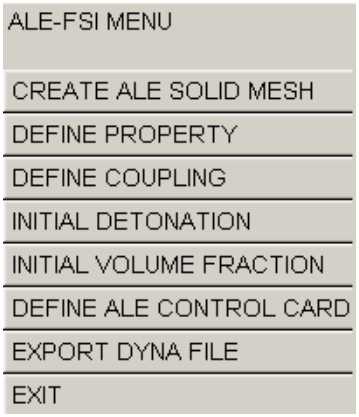
1. VPG prompts:
   >DELETE

2. Once the crash model is selected, it is highlighted. VPG prompts:
   >DO YOU WANT TO DELETE CRASH MODEL. (Y/N)
   - If YES is selected, the model will be deleted.

7.8 ALE-FSI MODULE

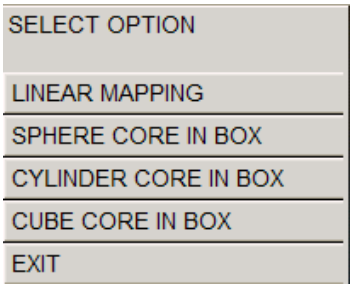
VPG's ALE/FSI modules are used for fluid and solid coupling simulations. It has following features:

- Simulate structure interaction with high energy explosion
- Parametric Soil/Fluid/Air ALE mesh generation
- Automated ALE material, section, EOS definition
- Easy FSI coupling
- Coupled with SPH elements
- Organized ALE control definition
- Export complete LS-DYNA input file. No editing required
- Creates standard meshes, sets up ALE zones, defines the coupling and material properties
- Defines control cards and exports the model



7.8.1 CREATE ALE SOLID MESH

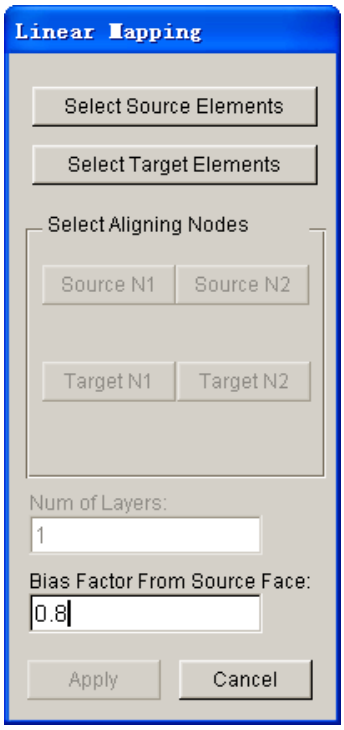
This option helps user to define standard ALE zones, and quickly sets up ALE Meshes and Explosive Source Models

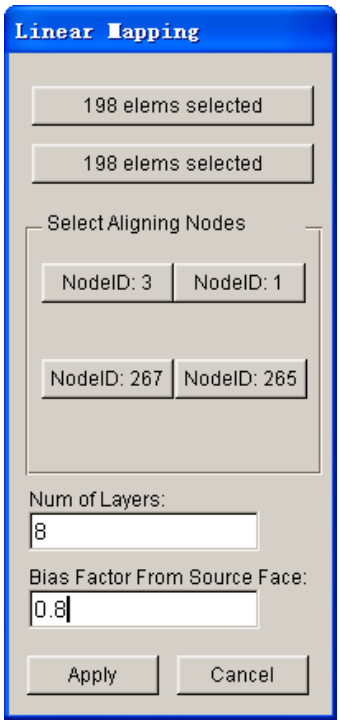


A detailed description of each function is given in the following sections.

7.8.1.1 LINEAR MAPPING

Description: This function allows the user to generate solid elements model by linear mapping.



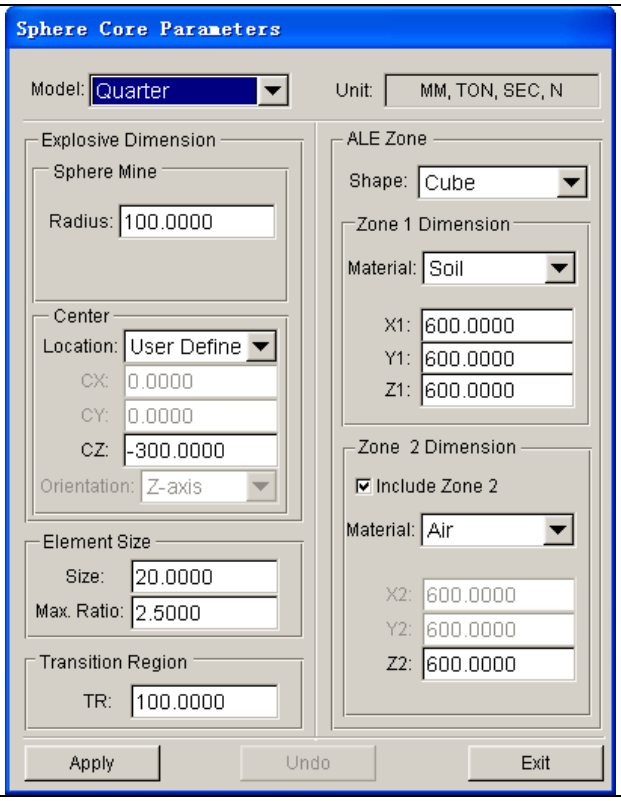


Usage:

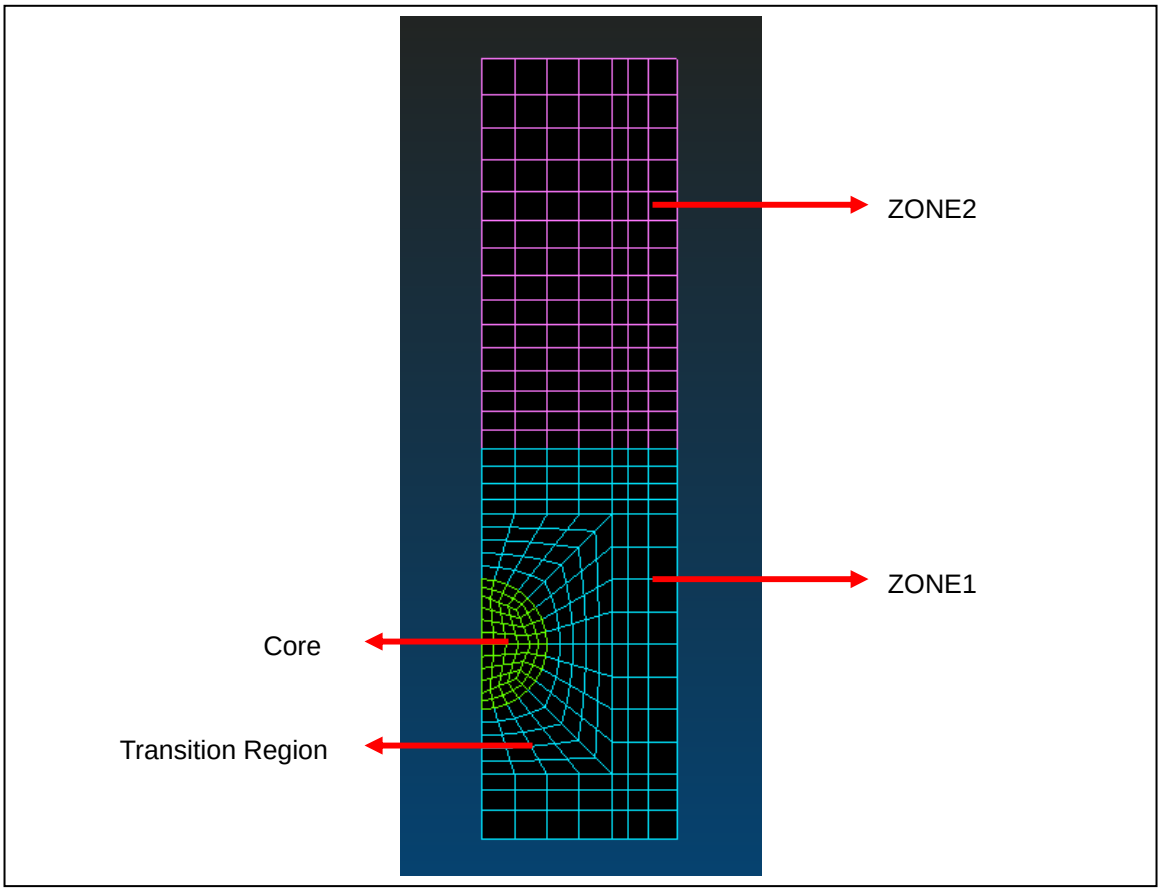
1. Select source elements and target elements, and the numbers of both elements should be the same.
2. The distributions of node locations are similar and VPG will automatically look for the aligning nodes in the model, sometimes, user needs to manually select the aligning nodes from source and target elements.
3. Set the number of layers between source elements and target elements, and the bias factor from source face; click the Apply button to generate solid elements by linear mapping.

7.8.1.2 SPHERE CORE IN BOX

Description: the generated explosive models are sphere fluid and solid coupling models. During the process, the materials, the element properties and the state equation of each part in the model will all be defined automatically.

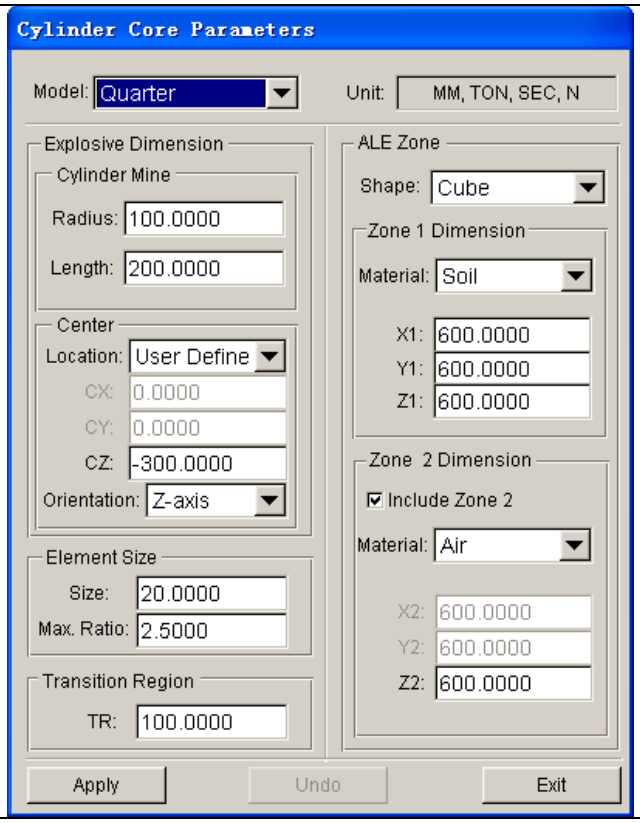


Sphere in box



7.8.1.3 CYLINDER CORE IN BOX

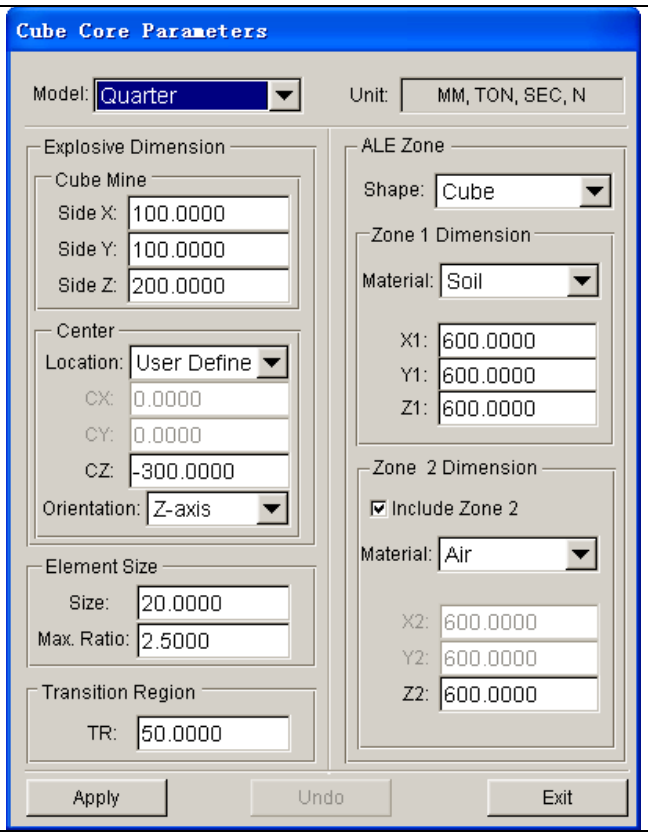
Description: the generated explosive models are cylinder fluid and solid coupling models.



Cylinder in box

7.8.1.4 CUBE CORE IN BOX

Description: the generated explosive models are cube fluid and solid coupling models.



Cube in box

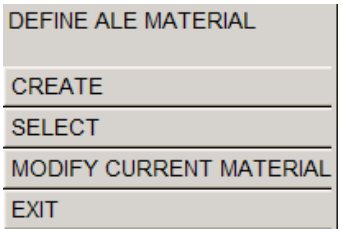
7.8.2 DEFINE PROPERTY

Description: this function allows the user to redefine the materials of explosive and medium in the model, and the properties of state equation.

Usage:

1. VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART

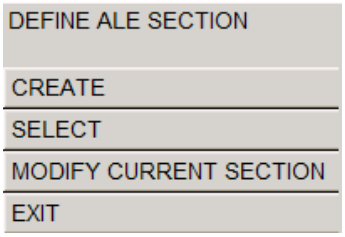
2. After a part is selected, VPG prompts:
   > SELECT OPTION



- **CREATE**, a new material will be created for selected part
- **SELECT**, a material will be selected for selected part
- **MODIFY CURRENT MATERIAL**, user can modify current material of selected part.
- **Click EXIT**, exit material definition.

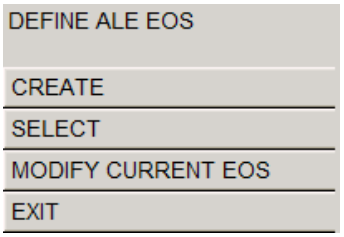
3. After material is defined, VPG prompts:

> SELECT OPTION



- **CREATE**, a new section will be created for selected part
- **SELECT**, a section will be selected for selected part
- **MODIFY CURRENT SECTION**, user can modify current section of selected part.
- **EXIT**, exit section definition.

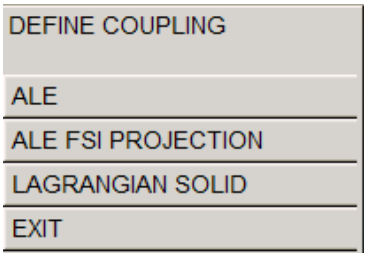
4. After section is defined, VPG prompts:
   > SELECT OPTION



- **CREATE**, a new EOS will be created for selected part
- **SELECT**, a EOS will be selected for selected part
- **MODIFY CURRENT EOS**, user can modify current EOS of selected part.
- **EXIT**, exit EOS definition.

7.8.3 DEFINE COUPLING

Description: this function is used to define the fluid and solid coupling between explosive and medium in the model.

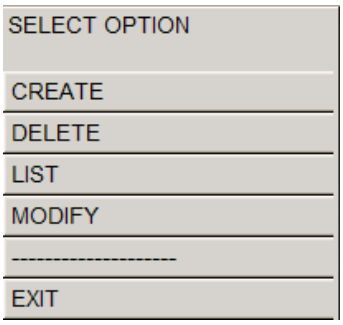


Usage:

- **ALE**, user can define keyword relative to *ALE
- **ALE FSI PROJECTION,** user can define coupling by keyword *ALE_FSI_PROJECTION
- **LAGRANGE SOLID,** user can define coupling by keyword *CONSTRAINED_LAGRANGE_IN_SOLID
- **EXIT,** exit coupling definition

7.8.4 INITIAL DETONATION

Description: this function is used to define the detonation parameters of explosives, including the detonation locations and detonation time.

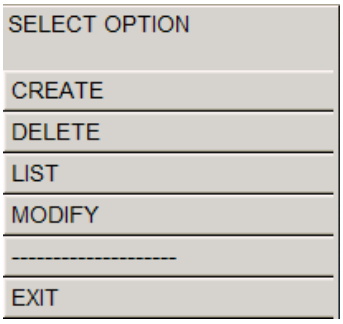


Usage:

- **CREATE**, define detonation card.
- **DELETE**, delete already created detonation card.
- **LIST**, list already created detonation card.
- **MODIFY**, modify already created detonation card.

7.8.5 INITIAL VOLUME FRACTION

Description: this function is used to define initial volume fraction of different materials in multi-material ALE elements.

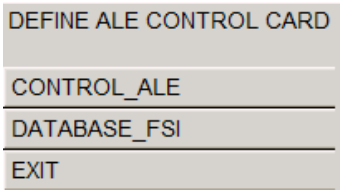


Usage:

- **CREATE**, define volume fraction card.
- **DELETE**, delete already created volume fraction card.
- **LIST**, list already created volume fraction card.
- **MODIFY**, modify already created volume fraction card.

7.8.6 DEFINE ALE CONTROL CARD

Description: this function is used to define ALE control options and output parameters.



Usage:

- **CONTROL_ALE**, define ALE coupling control card.
- **DATABASE_FSI**, ASCII file “dbfsi” definition, user can define, list and remove this file definition here
- **EXIT**, exit ALE control definition

7.8.7 EXPORT DYNA FILE

Description: this function is used to export the files submitted for dyna operation.

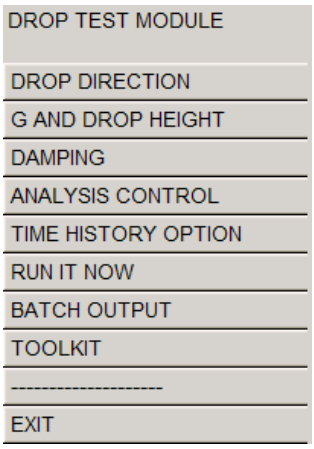
7.9 DTM MODULE

This DTM module includes functions DROP TEST, PRESSURE TEST, VIBRATION TEST and SHOCK TEST. These functions are mainly for reliability test for electric products and consumer products.

7.9.1 DROP TEST

DTM (drop test module) module is used for the drop analysis, especially the analysis on electronic products and consumer products. DTM module can automatically calculate the center of gravity, define and modify the drop direction, define the drop height, initial analysis height, drop acceleration and drop damping, and flexibly select the drop reference point/node and output control options. It can also batch output the drop analysis files of multiple heights. The floor and rigid floor required by the drop test, as well as the contact definition of drop models will be completed by DTM module automatically.

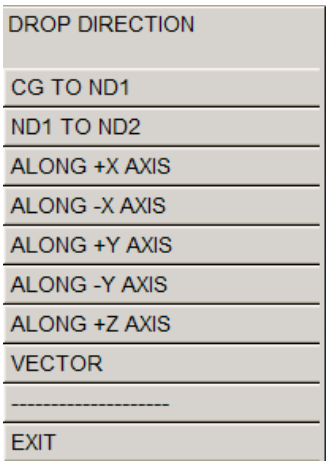
DTM module can quickly create simulation models for drop and impact tests in accordance with the process guidance. With various drop and impact test simulations, it can provide reliable reference for the product design.



A detailed description is given in the following sections.

7.9.1.1 DROP DIRECTION

Description: VPG module will automatically calculate the center of gravity location for the model when entering this menu. The user can adjust the drop direction after entering this menu. The model will rotate the view according to the user’s definition.

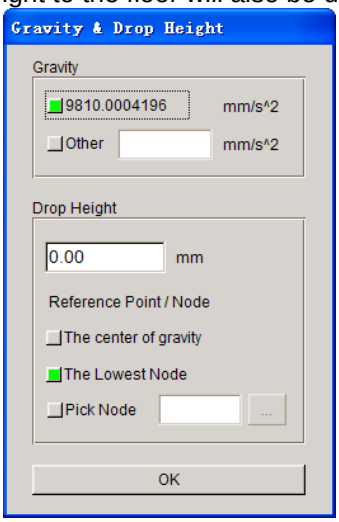


The default drop direction is along –Z axis. The executable direction adjustment contains:
CG TO ND1: the direction from the center of gravity to the selected node

- **ND1 TO ND2**: the direction from node 1 to node 2
- **ALONG +X AXIS**: along +X axis in the current view
- **ALONG –X AXIS**: along -X axis in the current view
- **ALONG +Y AXIS**: along +Y axis in the current view
- **ALONG –Y AXIS**: along -Y axis in the current view
- **ALONG +Z AXIS**: along +Z axis in the current view
- **VECTOR:** along the defined vector direction

7.9.1.2 G AND DROP HEIGHT

Descriptions: this function allows the user to use the default gravity acceleration of 9810.0004196mm/s^2, or arbitrary drop acceleration. The three selections for the reference point/node of drop height are: the center of gravity, The Lowest Node and Pick Node. After entering the drop height, the height to the floor will also be determined accordingly.



- **G**: define drop acceleration
- **9810mm/s^2**: default drop acceleration
- **Other**: user defined drop acceleration
- **Drop Height**: set drop height value
- **Reference Point/Node**: select reference point/node to determine the drop height
- **The Center of Gravity**: select center of gravity as reference

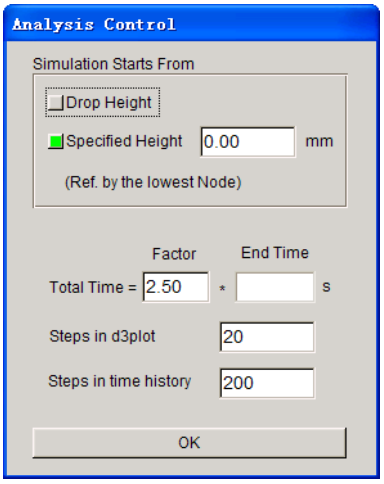
- **The Lowest Node**: select the lowest node in model as reference
- **Pick Node**: user pick a node as reference

7.9.1.3 DAMPING

Description: this function enables the user to define the damping during the model drop process. The damping can be constant, or be defined as the time function by curve.

7.9.1.4 ANALYSIS CONTROL

Description: this function allows the user to define the initial analysis height. If the initial analysis height is less than the drop height, DTM module will automatically calculate the initial velocity of the model at the beginning of the analysis, and calculate the total time spent when the model dropped from the initial analysis height to the floor.

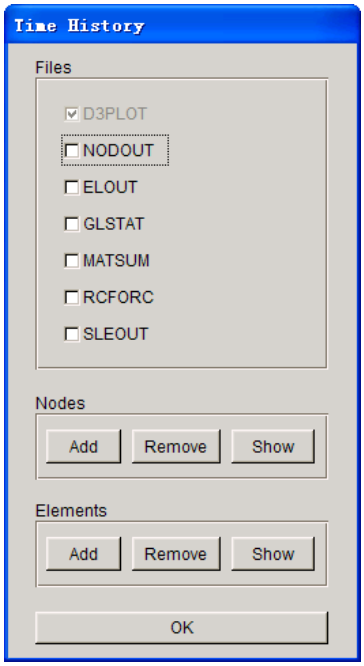


- **Simulation Start From**: define initial analysis height, note that this value should be less than or equal to the drop height.
- **Drop height**: take drop height as initial analysis height
- **Specified Height**: user define initial analysis height
- **Total Time**: time needed from initial analysis height to the floor
- **Steps in**: d3plot files steps during simulation
- **Steps in time**: ASCII files steps during simulation

7.9.1.5 TIME HISTORY OPTION

Description: this function is used to add and control the output file options. The user can select the desired nodes and elements to output the desired file. The types of the output files are:

- Animation file(d3plot)
- Node output file(nodout)
- Element output file(elout)
- Total energy message of the model(glstat)
- Material absorption capability (matsum)
- Resultant contact force during impact(rcforc)
- Contact energy message(sleout)

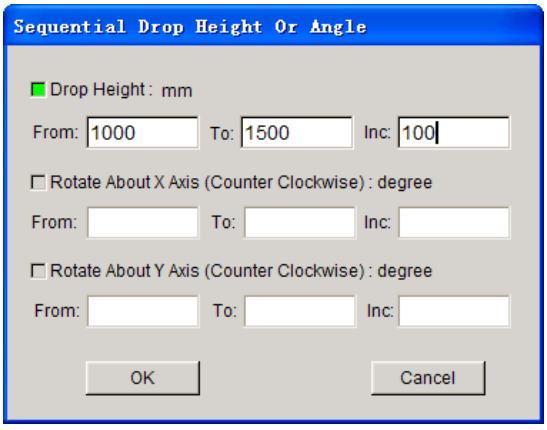


7.9.1.6 RUN IT NOW

Description: this function enables the user to output the analysis file and submit it to dyna immediately for simulation.

7.9.1.7 BATCH OUTPUT

Description: this function is used to output a series of calculation output files from different drop height or drop angle with other parameters stay unchanged. Multiple simulations can be completed at one time. The output files will be named as DTMdeck001.dyn, DTMdeck002.dyn, etc.



7.9.1.8 TOOLKIT

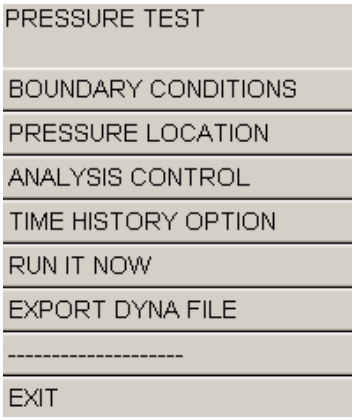
Description: this function is used for redo operation and setup.

Usage:

- RECALCULATE CG: recalculate the model's center of gravity.
- REMOVE DAMPING: remove the previously defined damping in DTM module.

7.9.2 PRESSURE TEST

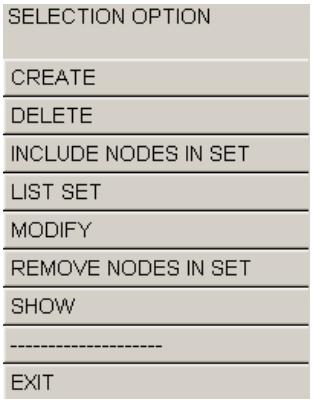
This Pressure Test can quickly create simulation models for pressure tests in accordance with the process guidance. For a Pressure Test process, users define a PUNCH with a certain diameter, then create a node set including interested test points. PUNCH will apply a pressure at the first node with a procedure of three phrases: Ramp up, Hold and Release. Then PUNCH moves to next node with the specified moving time period, and repeat the loading procedure until all node positions are tested.



A detailed description is given in the following sections.

7.9.2.1 BOUNDARY CONDITIONS

Description: This function helps to define SPCs for parts if needed.

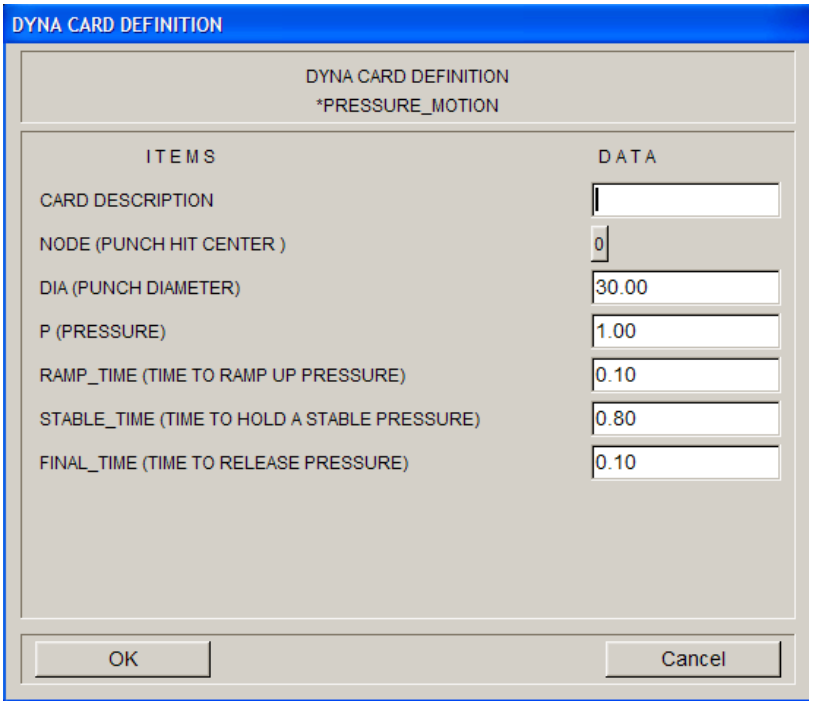


- **CREATE**: create SPC for selected nodes
- **DELETE**: delete SPC for selected node or node set.
- **INCLUDE NODES IN SET**: add nodes into the selected SPC node set; the new added nodes have the same SPC with nodes in the selected node set.
- **LIST SET**: show node set of SPC in a list
- **MODIFY**: modify the SPC for node or node set.
- **REMOVE NODES IN SET**: remove node from selected SPC node set.
- **SHOW:** show selected SPC node or node set in display window.

7.9.2.2 PRESSURE LOCATION

Descriptions: this function is used to define pressure for part punch, here a keyword

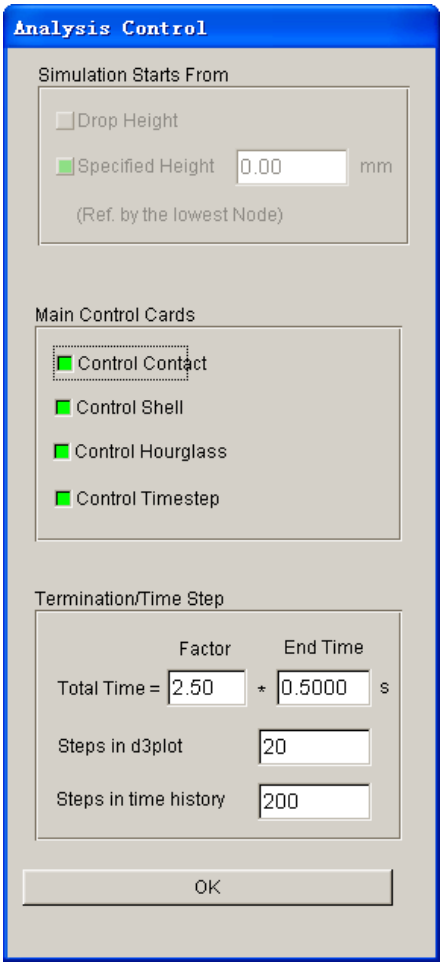
*PRESSURE_MOTION is created.



- **CARD DESCRIPTION**: name of this card.
- **NODE**: define pressure location, select one node as the punch hit center
- **DIA**: the diameter of punch cross section.
- **P**: the pressure value of punch.
- **RAMP_TIME**: the ramp up time for punch pressure at a node
- **STABLE_TIME**: the hold time for punch pressure at a node
- **FINAL_TIME**: the release time for punch pressure at a node

7.9.2.3 ANALYSIS CONTROL

Description: this function helps user to define control parameters such as *CONTROL_ CONTACT, *CONTROL_ SHELL etc. and also automatically calculates the termination time.

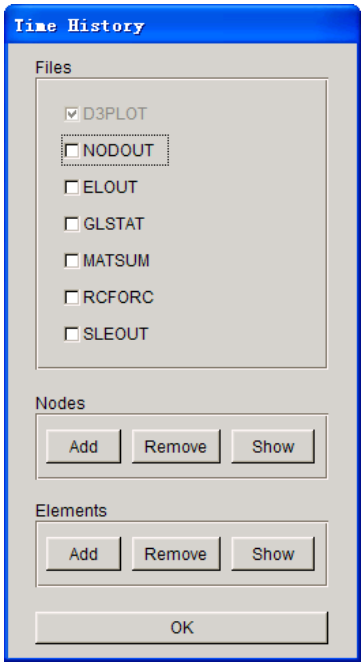


- **Simulation Start From**: define initial analysis height, note that this value should be less than or equal to the drop height.
- **Drop height**: take drop height as initial analysis height
- **Specified Height**: define initial analysis height
- **Main Control cards**: active control cards about contact etc.
- **Total Time**: time needed from initial analysis height to the floor
- **Steps in**: d3plot files steps during simulation
- **Steps in time**: ASCII files steps during simulation

7.9.2.4 TIME HISTORY OPTION

Description: this function is used to add and control the output file options. The user can select the desired nodes and elements to output the desired file. The types of the output files are:

- Animation file(d3plot)
- Node output file(nodout)
- Element output file(elout)
- Total energy message of the model(glstat)
- Material absorption capability (matsum)
- Resultant contact force during impact(rcforc)
- Contact energy message(sleout)



7.9.2.5 RUN IT NOW

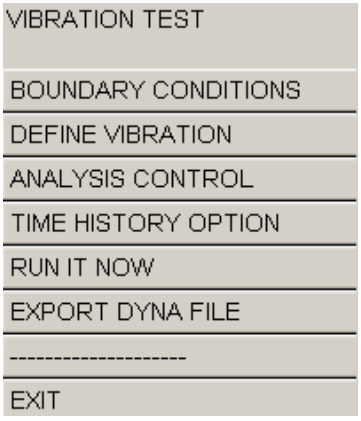
Description: this function enables the user to output the analysis file and submit it to dyna immediately for simulation.

7.9.2.6 EXPORT DYNA FILE

Description: this function exports the analysis file but do not run it now.

7.9.3 VIBRATION TEST

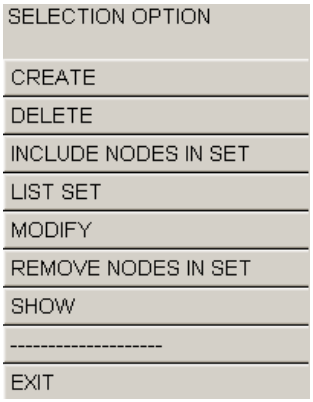
This Vibration Test can quickly create simulation models for vibration tests in accordance with the process guidance. For a Vibration Test process, Users define a vibration base with a certain diameter. A frequency domain PSD curve is created or read in; VPG converts it to a time domain motional curve. And a boundary motion is applied to the test Vibration base as specified, such as DOF, VAD. Typically a displacement motion along z-axle is applied to the base.



A detailed description is given in the following sections.

7.9.3.1 BOUNDARY CONDITIONS

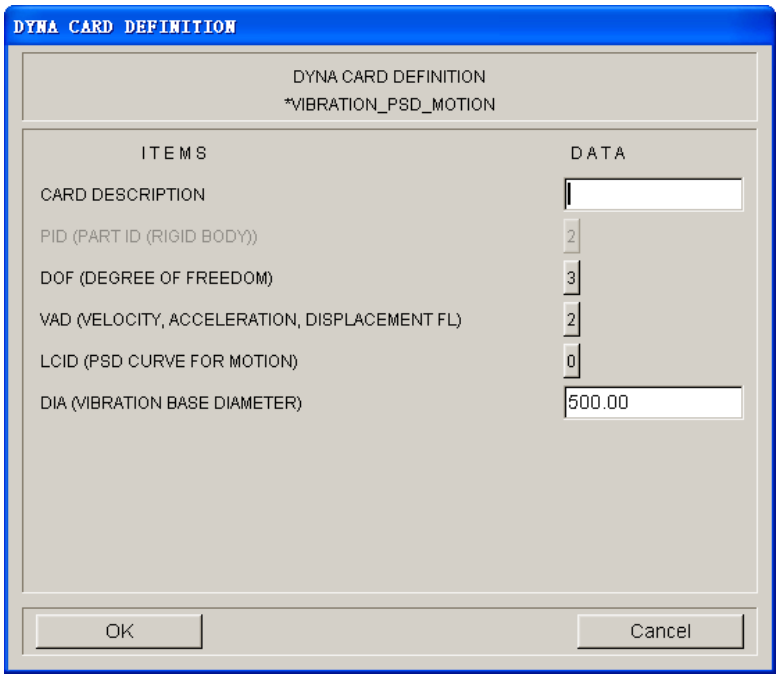
Description: This function helps to define SPCs for parts if needed.



- **CREATE**: create SPC for selected nodes
- **DELETE**: delete SPC for selected node or node set.
- **INCLUDE NODES IN SET**: add nodes into the selected SPC node set; the new added nodes have the same SPC with nodes in the selected node set.
- **LIST SET**: show node set of SPC in a list
- **MODIFY**: modify the SPC for node or node set.
- **REMOVE NODES IN SET**: remove node from selected SPC node set.
- **SHOW:** show selected SPC node or node set in display window.

7.9.3.2 DEFINE VIBRATION

Descriptions: this function is used to define vibration for part base, here a keyword *VIBRATION_PSD_MOTION is created.

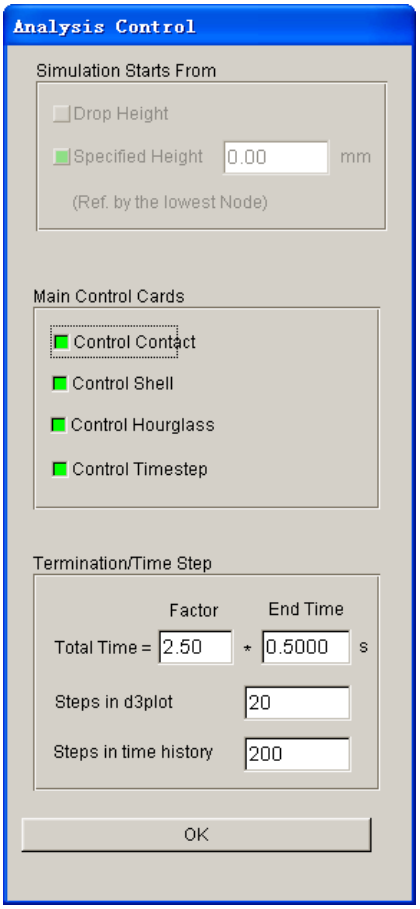


- **CARD DESCRIPTION**: name of this card.

- **PID**: part ID of the base.
- **DOF**: the degree of freedom of the base vibration.
- **VAD** set the vibration signal type.
- **LCID**: the vibration PSD value curve.
- **DIA**: the diameter of base cross section.

7.9.3.3 ANALYSIS CONTROL

Description: this function helps user to define control parameters such as *CONTROL_ CONTACT, *CONTROL_ SHELL etc. and also automatically calculates the termination time.

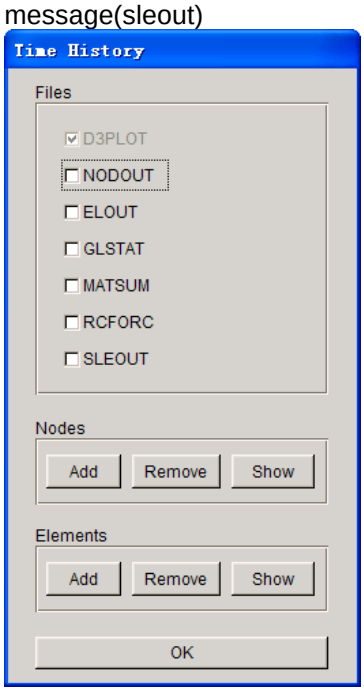


- **Simulation Start From**: define initial analysis height, note that this value should be less than or equal to the drop height.
- **Drop height**: take drop height as initial analysis height
- **Specified Height**: define initial analysis height
- **Main Control cards**: active control cards about contact etc.
- **Total Time**: time needed from initial analysis height to the floor
- **Steps in**: d3plot files steps during simulation
- **Steps in time**: ASCII files steps during simulation

7.9.3.4 TIME HISTORY OPTION

Description: this function is used to add and control the output file options. The user can select the desired nodes and elements to output the desired file. The types of the output files are:

- Animation file(d3plot)
- Node output file(nodout)
- Element output file(elout)
- Total energy message of the model(glstat)
- Material absorption capability (matsum)
- Resultant contact force during impact(rcforc)
- Contact energy message(sleout)



7.9.3.5 RUN IT NOW

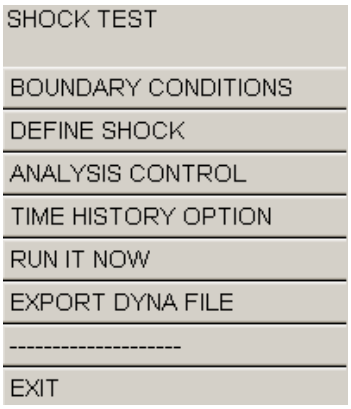
Description: this function enables the user to output the analysis file and submit it to dyna immediately for simulation.

7.9.3.6 EXPORT DYNA FILE

Description: this function exports the analysis file but do not run it now.

7.9.4 SHOCK TEST

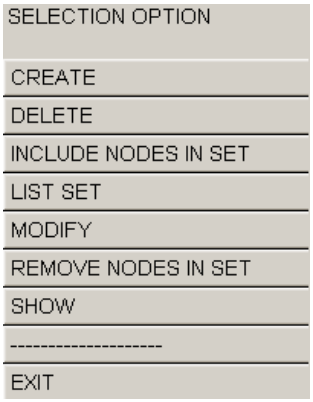
This Shock Test can quickly create simulation models for shock tests in accordance with the process guidance. For a Shock Test process, Users define a shock base with a certain diameter. A time domain curve is created or read in as a motional curve. And a boundary motion is applied to the test SHOCK base as specified, such as DOF, VAD. Typically an acceleration motion along z-axle is applied to the base.



A detailed description is given in the following sections.

7.9.4.1 BOUNDARY CONDITIONS

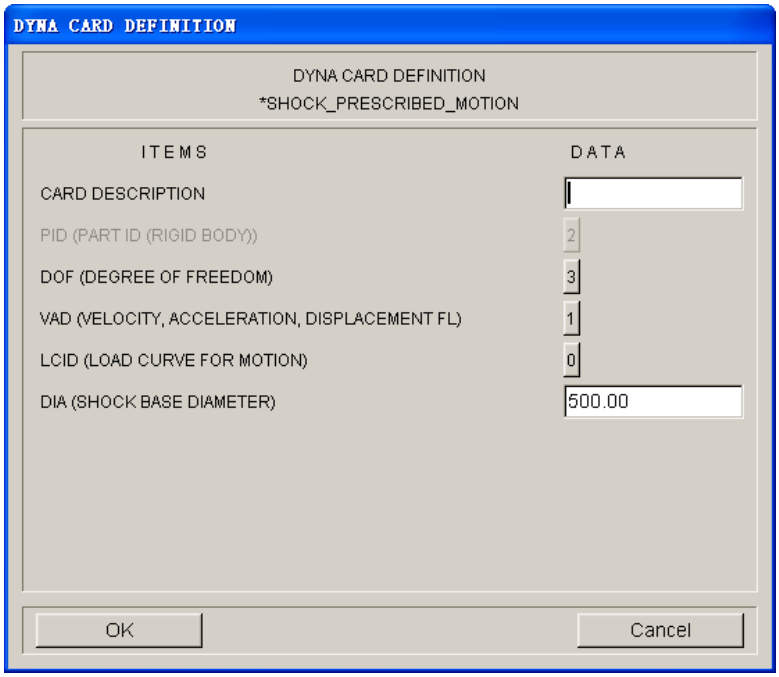
Description: This function helps to define SPCs for parts if needed.



- **CREATE**: create SPC for selected nodes
- **DELETE**: delete SPC for selected node or node set.
- **INCLUDE NODES IN SET**: add nodes into the selected SPC node set; the new added nodes have the same SPC with nodes in the selected node set.
- **LIST SET**: show node set of SPC in a list
- **MODIFY**: modify the SPC for node or node set.
- **REMOVE NODES IN SET**: remove node from selected SPC node set.
- **SHOW:** show selected SPC node or node set in display window.

7.9.4.2 DEFINE SHOCK

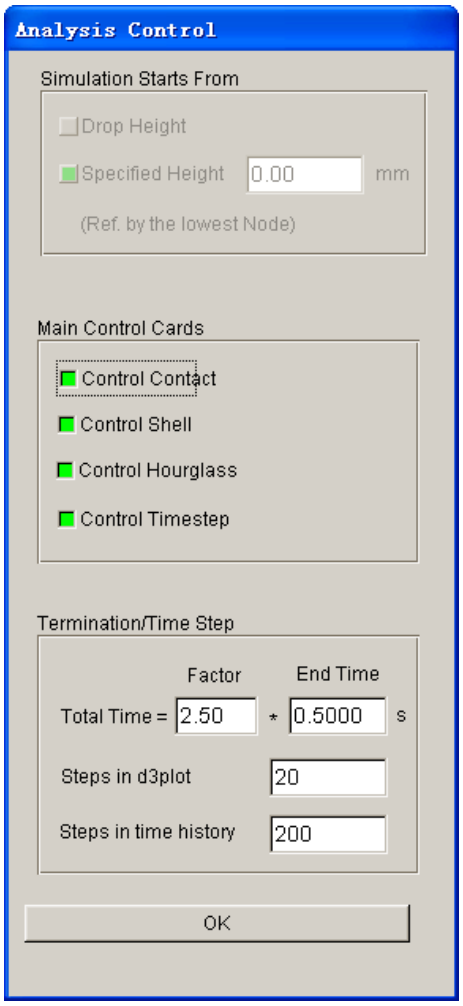
Descriptions: this function is used to define shock for part base, here a keyword *SHOCK_PRESCRIBED_MOTION is created.



- **CARD DESCRIPTION**: name of this card.
- **PID**: part ID of the base.
- **DOF**: the degree of freedom of the base shock.
- **VAD** set the shock signal type.
- **LCID**: the shock value curve in time region.
- **DIA**: the diameter of base cross section.

7.9.4.3 ANALYSIS CONTROL

Description: this function helps user to define control parameters such as *CONTROL_ CONTACT, *CONTROL_ SHELL etc. and also automatically calculates the termination time.

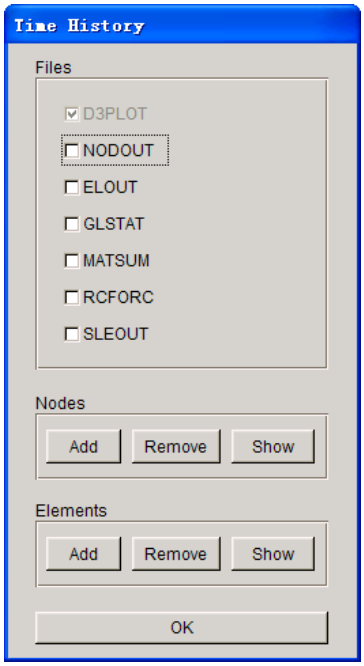


- **Simulation Start From**: define initial analysis height, note that this value should be less than or equal to the drop height.
- **Drop height**: take drop height as initial analysis height
- **Specified Height**: define initial analysis height.
- **Main Control cards**: active control cards about contact etc.
- **Total Time**: time needed from initial analysis height to the floor
- **Steps in**: d3plot files steps during simulation
- **Steps in time**: ASCII files steps during simulation

7.9.4.4 TIME HISTORY OPTION

Description: this function is used to add and control the output file options. The user can select the desired nodes and elements to output the desired file. The types of the output files are:

- Animation file(d3plot)
- Node output file(nodout)
- Element output file(elout)
- Total energy message of the model(glstat)
- Material absorption capability (matsum)
- Resultant contact force during impact(rcforc)
- Contact energy message(sleout)



7.9.4.5 RUN IT NOW

Description: this function enables the user to output the analysis file and submit it to dyna immediately for simulation.

7.9.4.6 EXPORT DYNA FILE

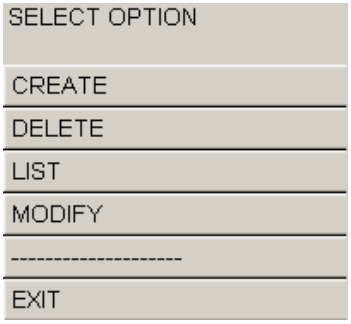
Description: this function exports the analysis file but do not run it now.

7.10 HEEDS

HEEDS is an optimization program based on NASTRAN, DYNA etc, which will help to find a best result and do much good to the design. HEEDS module in VPG supports to define HEEDS discrete set, distribution, variable, tag and output HEEDS file and relative DYNA file. For tagging, it now only supports DYNA material and section

7.10.1 DISCRETE SET

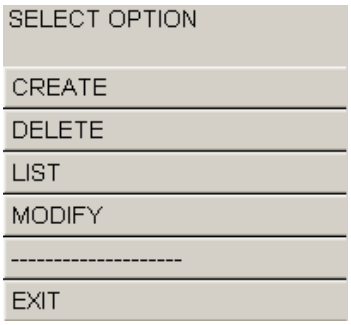
Description: This function helps user to define discrete set



- **CREATE**: create a discrete set
- **DELETE**: delete defined discrete sets.
- **LIST**: list defined discrete sets
- **MODIFY**: modify defined discrete sets.
- **EXIT**: exit and returns to the upper menu.

7.10.2 DISTRIBUTION

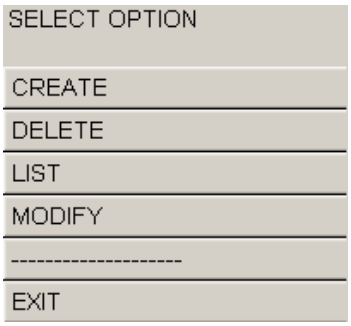
Description: This function helps user to define distribution



- **CREATE**: create a distribution.
- **DELETE**: delete defined distributions.
- **LIST**: list defined distributions.
- **MODIFY**: modify defined distributions.
- **EXIT**: exit and returns to the upper menu.

7.10.3 VARIABLE

Description: This function helps user to define variable



- **CREATE**: create a variable.
- **DELETE**: delete defined variables.
- **LIST**: list defined variables.
- **MODIFY**: modify defined variables.
- **EXIT**: exit and returns to the upper menu.

7.10.4 TAGGING

Description: This function helps user to tag defined variables to material or section parameters.

Usage:

1. The variable cards list pops up, and VPG prompts:
   > SELECT VARIABLE CARD TO TAG

2. After a variable is selected, VPG prompts:
   > SELECT OPTION
   MATERIAL
   - the defined materials list will pop up

   SECTION
   - the defined sections list will pop up

3. After a material or section is selected, their definition card will pop up, user now can

toggle on an option, and click OK, then the selected variable will be tagged to this option successfully.

7.10.5 EXPORT HEEDS FILE

Description: This function helps user to export DYNA input file; a HEEDS input file with the same name will be exported simultaneously.

Usage:

1. The export file dialog pops up and VPG prompts user to enter the DYNA input file name,

2. After input file name is entered, DYNA INPUT FILE OPTIONS dialog pops up, now user can input some information such as analysis title etc.
   - click OK, the DYNA input file is exported, and a HEEDS input file with the same name will be exported simultaneously.

7.10.6 THICKNESS OPTIMIZATION

Description: This function guides user to define parameters for thickness optimization.

Usage:

1. VPG prompts user to select a part from the part list,

2. After a part is selected, VPG prompts:
   > ENTER MINIMUM, MAXIMUM THICKNESS FOR SELECTED PART (MINIMUM > 0.0)

3. After the minimum and maximum thickness is entered, VPG prompts:
   > ENTER RESOLUTION FOR THICKNESS RANGE (RESOLUTION > 0.0)

4. After the resolution is entered, VPG prompts:
   > ENTER MAXIMUM Von Mises STRESS (> 0.0)

5. After maximum stress is entered, VPG prompts:
   > ENTER MEMORY SIZE (NUMBER OF WORDS) TO RUN DYNA JOB (> 0)

6. After memory size is entered, VPG prompts:
   > ENTER NUMBER OF EVALUATION (> 0)

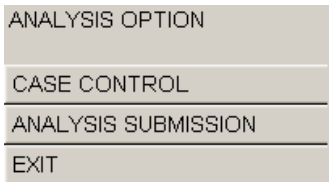
7. After number of evaluation is entered, VPG prompts to export input file to run.

Chapter

8

Chapter 8: ANALYSIS EXECUTION

VPG allows the user to define CASE CONTROL ANALYSIS, or to submit an analysis for execution directly from the VPG interface using the functions found in the ANALYSIS SUBMISSION menu.



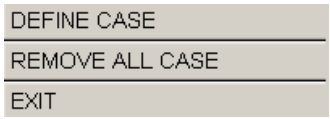
The CASE CONTROL option will lead user to define each case, user can select BOUNDARY CONDITIONS and LOADS defined in database for each needed case, all these cases share the same finite elements model, so user doesn't need to define B.C and LOADS in several databases, which will save user much time and energy.

The ANALYSIS SUBMISSSION Menu contains functions that create the necessary Control Cards and Analysis Parameters to specify a complete analysis model file. The ANALYSIS Menu allows users to define what type of analysis results will be generated from the analysis through the definition of DATABASE cards.

VPG provides a command to write and execute the analysis model from the ANALYSIS SUBMISSION menu, or alternately the option to export a file for later execution.

8.1 CASE CONTROL

The CASE CONTROL menu allows the user to define or delete case control.



8.1.1 DEFINE CASE

Description: this function allows the user to define the cases, all cases definition will be finished in the following pop-up window. All of the *BOUNDARY_SPC_, *BOUNDARY_PRESCRIBE_MOTION_, *LOAD_NODE_, *LOAD_SHELL and *LOAD_SEGMENT cards defined in database will be listed in the window. Users select needed for each case.

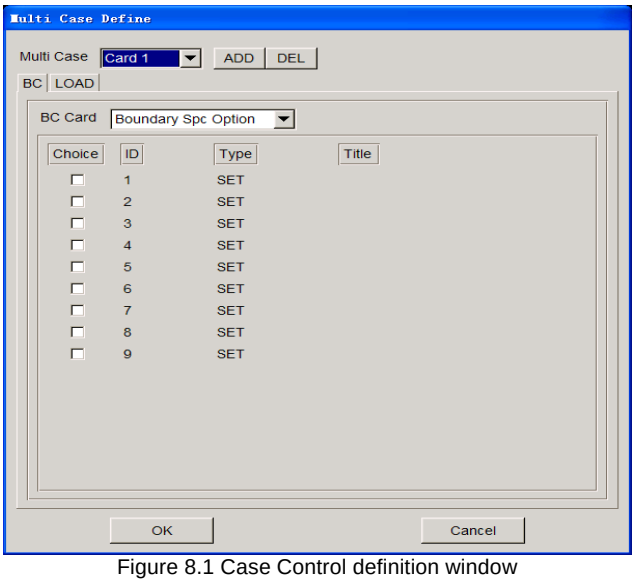


Figure 8.1 Case Control definition window

- **ADD**

A new case will be created and set as current case.

- **DEL**

Current case will be deleted, and the next case will be set as current case

- **OK**

Cases definition finished and exit.

- **CANCEL**

No cases modification will be done and exit.

8.1.2 REMOVE ALL CASE

Description: this function deletes all already defined cases.

8.2 ANALYSIS SUBMISSION

The **ANALYSIS SUBMISSION** menu allows the user to interactively run the VPG solver (LS-DYNA) from the Graphical User Interface window. The option buttons in the ANALYSIS

SUBMISSION window (CONTROL CARD, BOUNDARY CARDS, CONSTRAINT CARDS, etc.) provide access to the necessary functions for completing a DYNA input file. The toggle button at the bottom of the window, RUN ANALYSIS, writes an input file and submits the job for analysis when the user selects **OK**. The WRITE INPUT FILE toggle writes an input file without submitting an analysis.

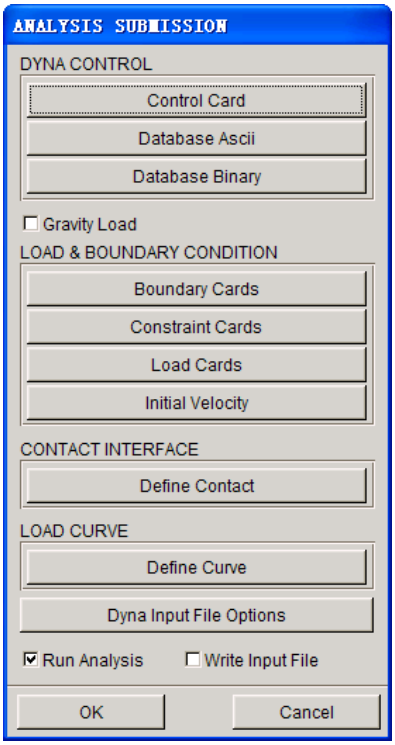


Figure 8.2 Analysis Submission Menu

The **DYNA INPUT FILE OPTIONS** button displays a window (see Figure 8.3) allowing the user to set the analysis unit for the job, the input file name, the analysis title, the engineer's identification, the project name, and the parts to be included.

8.2.1 CONTROL CARDS (LS-DYNA)

Description: Control Cards define simulation control parameters for various entities in the model. The values of these parameters are initially defined using the LS-DYNA default values. The user may modify any of these parameters to acceptable values.

All Control Cards defined in LS-DYNA version 971 are accessible through this menu. The use of the Control Card Menu is described in Section 6.14.4 (DYNA MISCELLANEOUS MENU).

8.2.2 DATABASE CARDS

Description: LS-DYNA models may produce various results data used for the study of the simulation and to assess the quality of the analysis. VPG allows users to define both the ASCII and BINARY databases that may be output from LS-DYNA. This is described in Section 6.14.6 (DYNA MISCELLANEOUS MENU) of this manual.

8.2.3 GRAVITY LOAD

Description: By selecting the GRAVITY LOAD option, VPG will automatically create a GRAVITY card with the appropriate acceleration of gravity for the unit system selected for the model.

8.2.4 BOUNDARY CARDS

Description: Boundary Cards allow the user to define boundary conditions on the model. Selecting BOUNDARY CARDS on the ANALYSIS menu directs the user to the BOUNDARY CARDS menu in the pre-processing BOUNDARY CONDITIONS menu (See Section 6.8 of this manual).

8.2.5 CONSTRAINT CARDS

Description: Constraint Cards allow the user to define constraint conditions on the model. Selecting CONSTRAINT CARDS on the ANALYSIS menu directs the user to the CONSTRAINT CARDS menu in the pre-processing BOUNDARY CONDITIONS menu (See Section 6.8 of this manual).

8.2.6 LOAD CARDS

Description: Load Cards allow the user to define loading conditions on the model. Selecting LOAD CARDS on the ANALYSIS menu directs the user to the LOAD CARDS menu in the pre-processing BOUNDARY CONDITIONS menu (See Section 6.8 of this manual).

8.2.7 INITIAL VELOCITY

Description: Selecting INITIAL VELOCITY allows the user to define initial conditions on the model. Selecting INITIAL VELOCITY on the ANALYSIS menu directs the user to the INIT. COND. CARDS menu in the pre-processing BOUNDARY CONDITIONS menu (See Section 6.8 of this manual).

8.2.8 DEFINE CONTACTS

Description: DEFINE CONTACTS allows the user to create, apply and modify contact interfaces within an LS-DYNA model. By selecting DEFINE CONTACTS on the ANALYSIS menu, VPG directs the user to the CONTACT INTERFACE menu in the pre-processing menu (See Section 6.13 of this manual).

8.2.9 DEFINE CURVE

Description: The DEFINE CURVE function on the ANALYSIS menu allows users to define load curves for use in LS-DYNA analyses. This function is described in detail in Section 10.4.8 of this manual.

8.2.10 DYNA INPUT FILE OPTIONS

Description: This function allows the user to define the analysis unit and the portion of the model that is to be exported into an LS-DYNA input file. The user may also add comments to the model describing the Analysis Title, Engineer's name, and the Project Name.

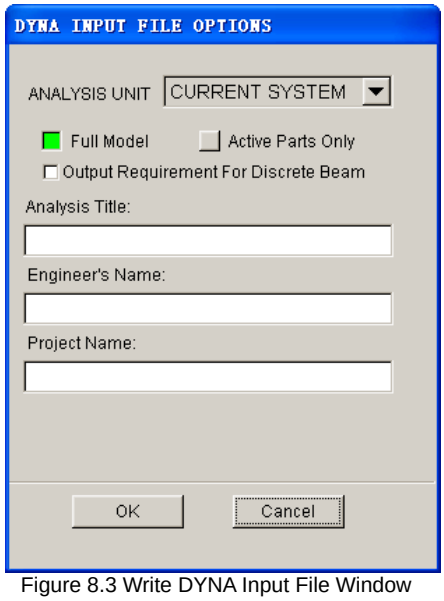


Figure 8.3 Write DYNA Input File Window

8.2.11 RUN ANALYSIS

Description: The user may elect to execute the analysis directly from within the VPG user interface. By selecting this option prior to selecting the OK button, the user will generate a model file according to the defined DYNA INPUT FILE OPTIONS. If the user wishes to create an LS-DYNA input file for later execution, the WRITE INPUT FILE option should be selected.

The user must have LS-DYNA and a valid license for this option to execute properly.

An execution window will be opened and an interactive LS-DYNA analysis will be started. This will not affect the operation of VPG.

8.2.12 WRITE INPUT FILE

Description: The WRITE INPUT FILE option allows the user to write an LS-DYNA input file to be executed at a later time outside of VPG. If the user wishes to automatically execute the simulation from within VPG, the RUN ANALYSIS option should be selected prior to selecting the OK button.

Chapter

9

Chapter 9: Post Processing

9.1 INTRODUCTION

The **POST** function in VPG is a general post-processor for all ETA software, including DYNAFORM, VPG, and FEMB. POST is an independent application module that is started when the user selects POST from the main menu.

The POST module quickly post-processes result data of finite element analyses, including the real-time animation of stresses, strain, energy, displacements, and time history curves. Its fully dynamic allocation of memory optimizes system resources, allowing for unlimited model sizes.

9.2 GENERAL OVERVIEW

The POST application interface varies slightly from the preprocessing user interface. It allows the user to quickly access most functions at any time during the session. The icon bar and menu bar functions behave in the same manner as the preprocessing functions.

eta/Post breaks the screen into six distinct regions. The regions are used to provide input or display messages to the user. The six regions are illustrated and described in Figure 9.1.

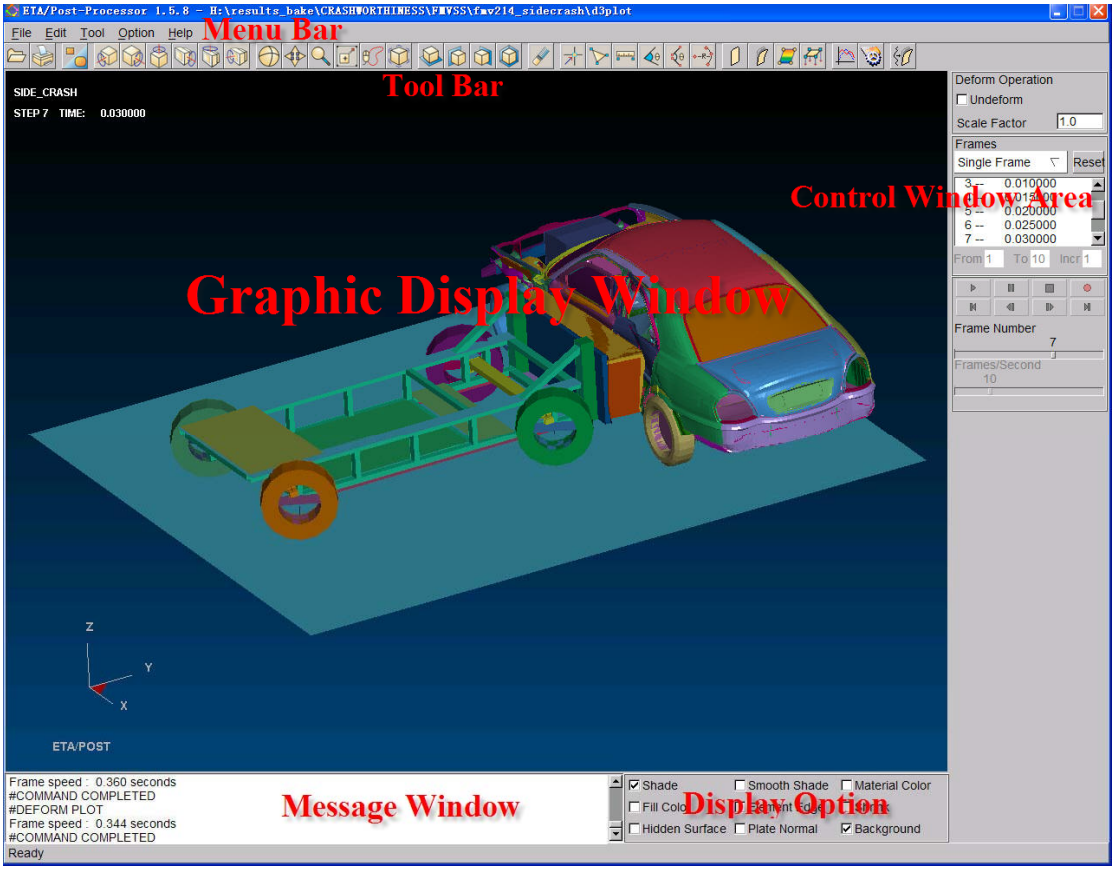


Figure 9.1 eta/Post interface window

GRAPHIC DISPLAY WINDOW
Models and graphs are displayed in this window.

MENU BAR
Commands and options are displayed in this area.

ICON BAR
Provides the user easy access to the commonly used functions of the program.

CONTROL WINDOW AREA
Once the user selects a command from the MENU BAR or ICON BAR, a corresponding dialog window with the appropriate functions is displayed in this area.

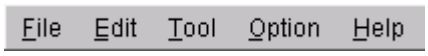
DISPLAY OPTIONS
The options in this group are used to control the model displayed in the graphic display window. These options are always displayed and can be used at any time during an eta/Post session.

PROMPT AREA
eta/Post displays comments and messages to the user.

The Graphic User Interface (GUI) is designed for easy access to all the functions for post-processing DYNA and Nastran results. This GUI is unified for both Windows and UNIX/LINUX platforms. The user activates the functions via mouse click.

9.3 MENU BAR

The menu bar contains the FILE, EDIT, TOOL and OPTION menus. All the menus are designed in a logical and efficient way to minimize number of mouse clicks and operations. The FILE menu contains functions that control the input and output of data files, the EDIT menu contains functions that allow the user to alter the model/graph and the VIEW menu contains functions that alter the display. Each of these menus branches into various submenus and functions.



FILE MANAGER	Imports and exports data from eta/Post (see Section 9.7)
EDIT	Organizes the functions for model operation.
TOOLS	eta/Post's "tool kit."
OPTION	Provide options in the graphics display window.
HELP	Displays the program information and phone number for technical support.

The menus and functions will be described in the related sections.

9.4 ICON BAR

The icon bar is designed to give the user easy access to the most commonly used functions in eta/Post. Some of the functions represented in the icon bar are also located in the different menus. The user may simply click these icons to activate the functions instead of navigating through the menus.

OPEN
Opens a result or model file.

PRINT

Creates a postscript file of the display area and sends it to the printer (default) or writes the image to a file. Prior to printing, the printer must be initialized to accommodate the print out from eta/Post software.

PART ON/OFF

This function enables the user to turn the selected part(s) on/off. Once the function is selected, the PART ON/OFF dialog window is displayed shown as Figure 9.2.

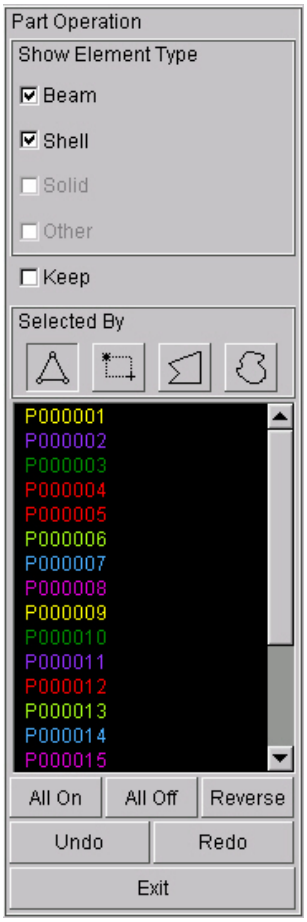


Figure 9.2 PART ON/OFF Window

Note: When PART ON/OFF dialog window is displayed, only VIEW OPREATION functions are enabled to use, other functions are disabled.

User can select part to turn on or off by picking the part name in the part list. There are several other methods to turn part on or off as described below.

SELECT BY

PICK ELEMENT

The part including the selected element is selected. The element will be highlighted as the

cursor moves over the part.

SELECT BY DRAG WINDOW
This function allows the user to drag a window around the desired parts, All parts including the selected elements are selected.

SELECT BY MULTI-REGION
This function allows the user to select a multi-point region, all parts including the selected elements are selected. Points are selected by clicking the mouse in the Graphic Display area.

SELECT BY FREE HAND
This function allows the user to select a region by a free hand, all parts including the selected elements are selected. The user depresses the left mouse button and moves the mouse to select the desired region.

SELECT BY ELEMENT TYPE
This function allows the user to turn selected element types on/off so that they are shown or no longer shown in the window list and in the GRAPHIC DISPLAY window.

BEAM
All parts with the beam element are removed from the part list if the BEAM is toggled off. And all parts with the beam element appear in the part list if the BEAM is toggled on.

SHELL
All parts with the shell element are removed from the part list if the SHELL is toggled off. And all parts with the shell element appear in the part list if the SHELL is toggled on.

SOLID
All parts with the solid element are removed from the part list if the SOLID is toggled off. And all parts with the solid element appear in the part list if the SOLID is toggled on.

OTHER
All parts with the line are removed from the part list if the OTHER is toggled off. And all parts with the line appear in the part list if the OTHER is toggled on.

KEEP

If **KEEP** is selected, the selected parts are highlighted and are labeled with asterisk on the Part List. When the user exits the function, only the selected parts remain displayed.

ALL ON

Turns all parts on and displays them on the screen.

ALL OFF

Turns all parts off from the GRAPHICS DISPLAY.

REVERSE
This function reverses the current on and off status of the parts in the model. The program updates the model display accordingly.

UNDO

Undo the last operation or part selection in Part ON/OFF function.

REDO

Redo the last operation in the part ON/OFF function.

EXIT

Closes the PART ON/OFF dialog window and exits the function.

VIRTUAL X ROTATION

The displayed model will dynamically rotate about the global X-axis when the cursor is moved up or down.

VIRTUAL Y ROTATION

The displayed model will dynamically rotate about the global Y-axis when the cursor is moved up or down.

VIRTUAL Z ROTATION

The displayed model will dynamically rotate about the global Z-axis when the cursor is moved up or down.

SCREEN X ROTATION

The displayed model will dynamically rotate about the screen X-axis when the cursor is moved up or down. The screen X-axis is the horizontal axis located at the center of the graphics display window.

SCREEN Y ROTATION

The displayed model will dynamically rotate about the screen Y-axis when the cursor is moved up or down. The screen Y-axis is the vertical axis located at the center of the graphics display window.

SCREEN Z ROTATION

The displayed model will dynamically rotate about the screen Z-axis when the cursor is moved up or down. The screen Z-axis is the normal direction of the screen, located at the center of the graphics display window.

FREE ROTATION

This function is a combination of Screen X and Screen Y rotation. Moving the mouse up/down manipulates the Screen X rotation. Moving the mouse left/right manipulates Screen Y rotation. Moving the mouse diagonally combines the rotations of both directions. Click the left mouse button to stop the rotation. The user may activate this function by pressing Control and left mouse button while moving the mouse. Release the mouse button to stop the rotation.

PAN

This command enables the user to translate the model on the screen by following the movement of the cursor. If the cursor is moved off the graphics display window, the cursor reappears at the

opposite edge of the window to continue the operation. Click the left mouse button to stop the pan operation. The user may activate this function by pressing Control and Middle mouse button while moving the mouse. Release the mouse button to exit the function.

CURSOR ZOOM
The user picks a point about which to zoom. The model is centered about this point and the user may move the cursor up to zoom in or down to zoom out. The user may activate this function by pressing Control and right mouse button while moving the mouse. Release the mouse button to exit the function.

Note: If the cursor is moved off the graphics display window in functions ROTATE, PAN or CURSOR ROOM, the cursor re-appears at the opposite of the window automatically to continue the operation.

WINDOW ZOOM
The user defines the corners of the zoom window by positioning the cursor on the display screen. The user presses and holds the left mouse button and drags the cursor diagonally until the desired window size is reached. Releases the left button, the area included in the window will be displayed in full screen.

FREE HAND ZOOM
The user defines a free region by pressing the left mouse button and dragging the cursor on the display screen to define the region. Releases the left button, the area included the region will be displayed in full screen.

FILL
Rescale the model to include all entities that are currently displayed. FILL automatically zooms in or out until the model fits the viewing area of the screen.

TOP VIEW
Automatically displays the model from the TOP or in the XY-plane.

Y-Z VIEW
Automatically displays the model from the RIGHT or in the YZ-plane.

X-Z VIEW
Automatically displays the model from the LEFT or in the XZ-plane.

ISOMETRIC VIEW
Automatically displays the model from the isometric plane (60-degree isometric).

CLEAR
Remove the highlighted entities from the screen.

IDENTIFY NODE

This function enables the user to identify any node by cursor selection (default) or key in node number. The program labels the node number of the selected node is highlighted on the screen and the outputs the coordinates in the message window. Figure 9.3 shows the CONTROL OPTION window when the function starts.

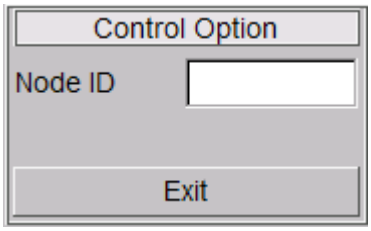


Figure 9.3 Identify Node Control Option Window

SELECT BY CURSOR (Default)
The program highlights the node that is nearest the cursor. Click the left mouse button when the desired node is traced. The selected node will be labeled with the node number.

KEY IN ID
This option allows user to identify the element by entering a node number. The user enters the node number and selects OK. The desired node will be highlighted and the node number displayed on the screen.

IDENDTIY ELEMENT

This function enables the user to identify any element, its part and its nodes by cursor selection (default) or key in ID number. The program will highlight the selected element with the element number on the screen and also display the element connectivity in the message window. The operation of this function is same as the IDENTIFY NODE function. Figure 9.4 shows the CONTROL OPTION window when the function starts.

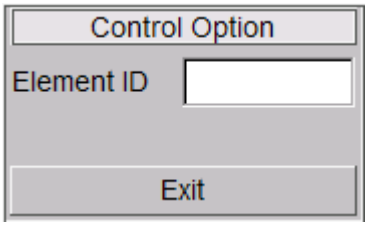


Figure 9.4 Identify Element Control Option Window

DISTANCE BETWEEN TWO NODES

This function calculates the distance between two nodes selected by cursor pick or key in node number. The program displays a CONTROL OPTION window as shown as Figure 9.5.

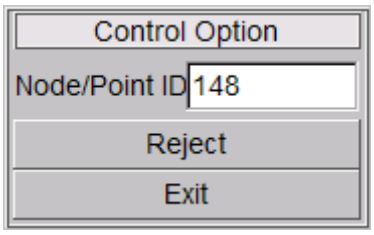


Figure 9.5. Measure distance Control Option window

The operation on SEELECT BY CURSOR and KEY IN ID is same as in IDENTIFY NODE function. REJECT LAST allows user to reject the last selected node. Once two nodes are selected, the program labels the distance and X, Y, Z components between the selected nodes on the screen and also outputs the information in the message window.

ANGLE BETWEEN THREE NODES

This function measures the angle between two vectors formed by three nodes. The first selected node defines the vertex of the angle. The program displays a CONTROL OPTION window that is common with the one in DISTANCE BETWEEN TWO NODES function. Once three nodes are selected, the program labels the angle at the vertex node and outputs the information in the message window.

ANGLE BETWEEN TWO LINES

This function is used to measure the angle between two lines that are composed by selected four nodes. Default setting is to select by mouse click or inputting node numbers. After selecting four nodes, program displays the angle at the two lines by selected four nodes and at the same time output information in the information window.

RAIDIUS BETWEEN THREE NODES

This function enables the user to measure the radius of an arc that is passing through three selected nodes by cursor selection (default) or key in node number. After selecting three nodes the program will display the radius on screen and output the coordinates of the center of the arc in message window.

GRAPH

This function is used to read ASCII file results, plot and operate curves**.**

AD2FEA

This function will transform ADAMS model to FEA model

NODOUT TO MODE

This function will do a FFT transformation to the ASCII file result.

9.4.1 POSTPROCESSING ICONS

The functions in the POSTPROCESS icon bar allow the user to access the functions to process the result files. There are two kinds of icon on the eta/Post, one is global icons, which control part on/off, views, and dynamic rotation. The second group of icons controls the type of post processing to be performed; deformed shape, stress, vector plots of graphing. Each of these are described in the following sections of this chapter.

Figure 9.6 Post Processing Icons

9.4.2 DISPLAY OPTIONS

Description: The DISPLAY OPTIONS window displays the current part and contains the following commonly used functions.

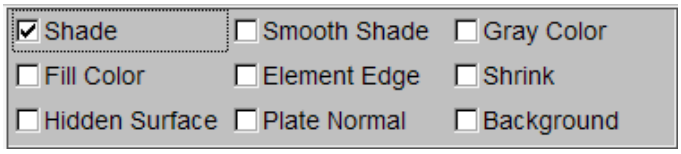


Figure 9.7 Display Options

9.4.2.1 SHADE (toggle)

Description: This command displays the elements as if they were illuminated by a light source. Elements that are not directly exposed to the light source are appropriately “shaded” to simulate the actual shading effect.

Eta/Post uses two methods for object shading: flat and smooth. Flat shading shades each polygon upon the intensity of the light over a series of polygons making the elements appear flat and angled.

9.4.2.2 SMOOTH SHADE (toggle)

Description: This function uses the Gourand shading method to make the model appears more smoothly. The feature angles between adjacent elements are smoothed by this shading method.

Note: The SMOOTH SHADE option is only available when the SHADE option is on.

9.4.2.3 GRAY COLOR (toggle)

Description: This function can only be used in SHADE mode. If the function is toggled on, the model will be plotted in gray color during deformed plot or animation deformation and the parts without stress/strain during contour plot or animation. If the parts are defined with material color in Part Attribute function, they will be shaded in the material color.

Note: The GRAY COLOR option is only available when the SHADE option is on.

9.4.2.4 FILL COLOR (toggle)

Description: This function toggles on/off the model in FILL COLOR mode. FILL COLOR fills the displayed elements with their designated part color.

9.4.2.5 ELEMENT EDGE (toggle)

Description: This function can only be used in FILL COLOR mode. The outline of the elements

is plotted in white when **options** is toggled on. The model can be displayed without the outline by toggling off the option.

Note: The ELEMENT EDGE is only available when the FILL COLOR or SHADE option is on**.**

9.4.2.6 SHRINK (toggle)

Description: SHRINK creates a plot with elements reduced in size by 20 percent. This option allows the user to toggle the function on/off.

9.4.2.7 HIDDEN SURFACE

Description: This function toggles on/off the model in HIDDEN SURFACE mode. HIDDEN SURFACE hides the elements behind the elements viewed from the user's point of view.

9.4.2.8 PLATE NORMAL (toggle)

Description: This function toggles plate normal on and off. The plate normal is shown with a vector drawn at the center of element and along the element's normal direction.

9.4.2.9 BACKGROUND (toggle)

Description: If this function is toggled on, the background color is set to white. Otherwise, default background color is black.

9.4.3 PRIORITY OF FUNCTIONS

Description: The functions in eta/Post are prioritized. Functions in Display Options can be accessed to at any time. The functions in Edit and Tool menu have the highest priority. If any of these functions is started, all other functions in eta/POST are disabled.

Control Window has the lowest priority. The user is enabled to access other functions when Control window is opened. Only functions in Edit menu, Tool menu and Display Option can be used during animation.

9.4.4 CONTROL WINDOW AND CONTROL OPTION WINDOW

Description: eta/Post incorporates two types of window, FUNCTION DIALOG WINDOW and CONTROL WINDOW. At the bottom of the FUNCTION DIALOG WINDOW there are buttons to execute, reject, reset the data or close the window. The functions of these buttons are listed below.

Usage:	**APPLY**	Executes the current function.
	CANCEL	Rejects the current operation.
	EXIT	Exits the current window
	OK	Accepts the data in the dialog box and forwards the user to the next step.
	UNDO	Rejects the last step of the operation.

RESET Restores the original set.

REJECT Rejects the previous selection.

9.5 FILE FORMAT

The protocol for naming files during an eta/Post session includes attaching suffixes to the file names that specify the file types. The appropriate file names are listed in the option area of the display screen.

1. DYNA result file (d3plot, d3drif, dynain, d3plotaa, d3plotint)
2. DYNA model file (*.dyn)
3. DYNA result file of eigenvalue analysis (d3eigv)
4. NASTRAN results file (*.pch, *.op2)
5. NASTRAN model file (*.nas, *.dat)
6. DYNAFORM/VPG/FEMB LINE DATA file (*.lin)

eta/Post can open DYNA result files to process the results directly. After the NASTRAN result file is loaded, the program will require loading the corresponding NASTRAN model file. The user may import LINE DATA, DYNA or NASTRAN model files.

9.6 CONFIGURATION FILE

The **etapost.config** file or initialization file controls the default setting of eta/Post. This file is located in the installation directory and can be edited via the text editor. The normal content of the etapost.config file is:

```
#ETA/Post User Configure File

[GRAPHIC ENGINE]
Z Buffer Bit = 16
Color Buffer Bit = 16
Edge Color = 255 255 255
Background Color = 0 0 0
Xor Plotter Styler = GDI
Light Source Type = LOCAL
Second Render = OFF
Debug = OFF

[WINDOW PARAMETER]
Layout Type = RIGHT

[PRODUCT PARAMETER]
Product Name = DYNAFORM
Extended GUI = ON

[DIRECTORY]
Home = F:\QA_Model\DF\test

[INDEX FILE]
File Name = etapost.idx
```

9.6.1 GRAPHIC ENGINE

Description: The parameters in this group are used to customize the OpenGL behavior with various graphics cards.

Usage:

Z buffer Bit Sets the bits of Z buffer when rendering a model, the default value is 16

Color buffer Bit	Setting the bits of color buffer when displaying a model, the default value is 16
Edge color	Setting the color of the edge of model when rendering, the default color is white.
Background color	Setting the background color of Graphic Display Window, the default color is black.
Xor Plotter Styler	Setting the Xor Plotter style, the default style is GDI.
Light Source Type	Setting the type of light source, the default type is local light source.
Second Render	Setting the second render on/off, the default type is off. NVIDIA Quadro FX series high performance graphic cards require this option set to ON in order to work properly.
Debug	Setting the mode of version debug on/off, the default mode is off.

9.6.2 WINDOW PARAMETER

Description: The parameter LAYOUT TYPE controls the location (LEFT or RIGHT) of the CONTROL WINDOW.

Usage: Default setting is on the RIGHT side of the screen. The user may change the parameter to LEFT to place the CONTROL WINODW on the left side of the screen.

9.6.3 PRODUCT PARAMETER

Description: The parameters in this group are used to customize the product parameters.

Usage:

Product Name	Setting the product of Pre-Processor of eta. There are two kinks of product, one is DYNAFORM, and another is VPG. The content of function icons in the Control Window is different between DYNAFORM and VPG. The default is DYNAFORM.
Extended GUI	Setting the extended GUI on/off, the default setting is on.

9.6.4 DIRECTORY

Description: This parameter records the directory of the latest opened file.

9.7 FILE MENU

Description: The options in this pull-down menu are used to open, save, import, export, and print related files. See Figure 9.8.

Figure 9.8 The File Menu

A detailed description of each function is given in the following sections.

9.7.1 OPEN

Description: This function displays the Open File dialog and allows the user to read result file or model file into eta/Post. The open file dialog is shown in Figure 9.9.

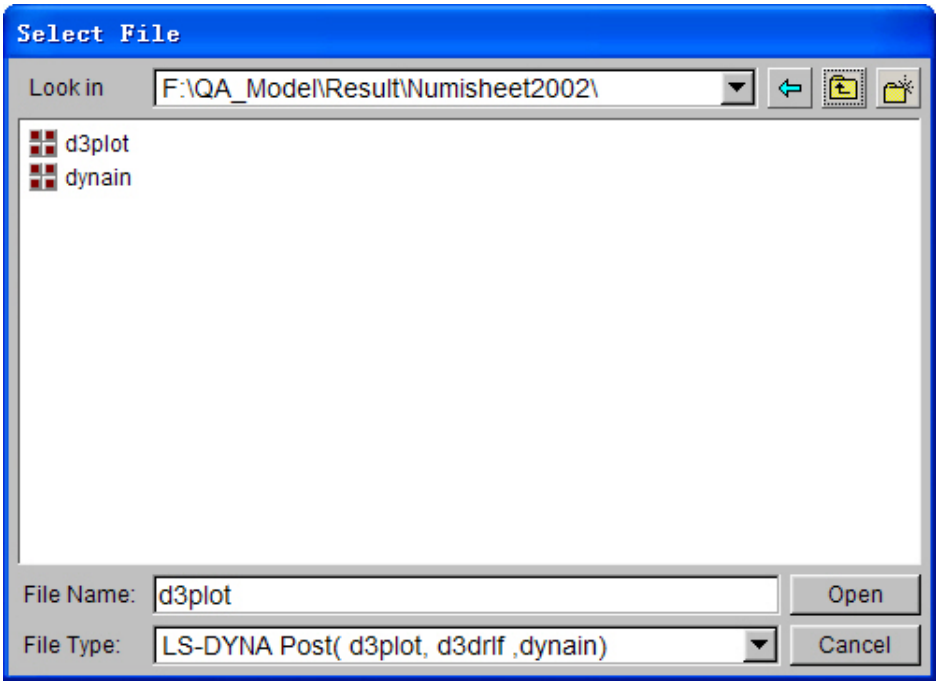


Figure 9.9 Select File window

Usage: To open result file or model data, select File Type that will be read in. Locate the desired file using the browser and click OPEN. The model win be displayed in the Graphic Display Window and will be ready for post processing.

Eta/Post supports LS-DYNA d3plot, d3eigv and d3drif, NASTRAN punch and output2 result files for post processing. The program will display the Select File window again for the user to select a Nastran model file after the Nastran result file is loaded. It is necessary to select a matching Nastran model in order to post-processing the result correctly.

9.7.2 IMPORT

Description: This function allows the user to import DYNAFORM/VPG/FEMB Line Data file for SKID MARK and BLANK OUTLINE functions. The Import File dialog is shown in Figure 9.10.

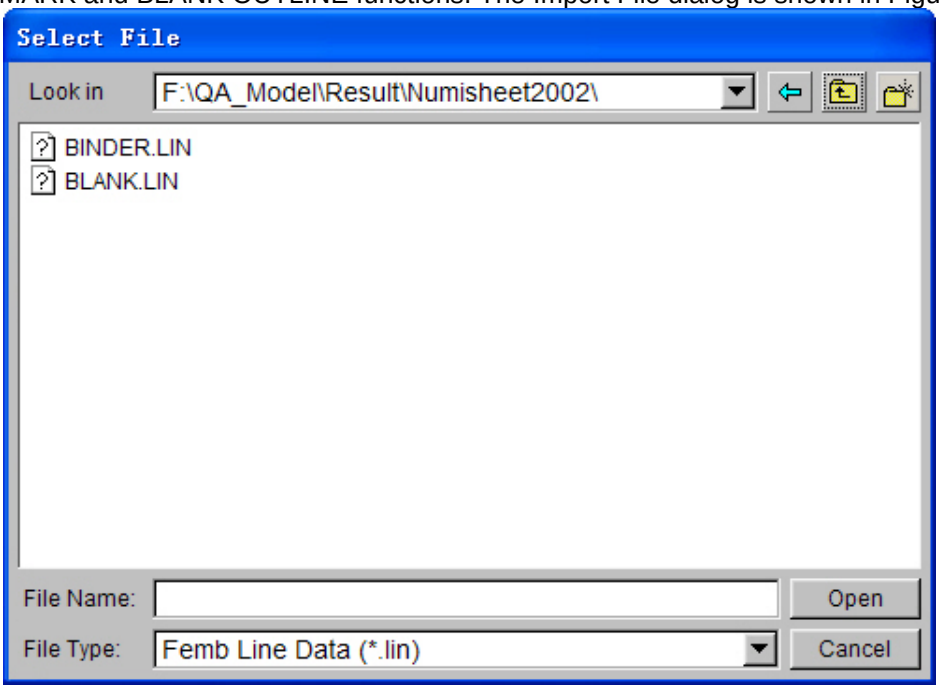


Figure 9.10 Import File window

User: User can select a DYNAFORM/VPG/FEMB Line Data file, then click OPEN button, or double click the desired file icon from the file name list to import it.

Note: IMPORT function is disabled until a d3plot file or NASTRAN punch/output2 file is read in.

9.7.3 EXPORT

Description: This function allows user to export the current model as NASTRAN file or DYNAFORM/VPG/FEMB line data file. The Export File dialog is shown in Figure 9.11.

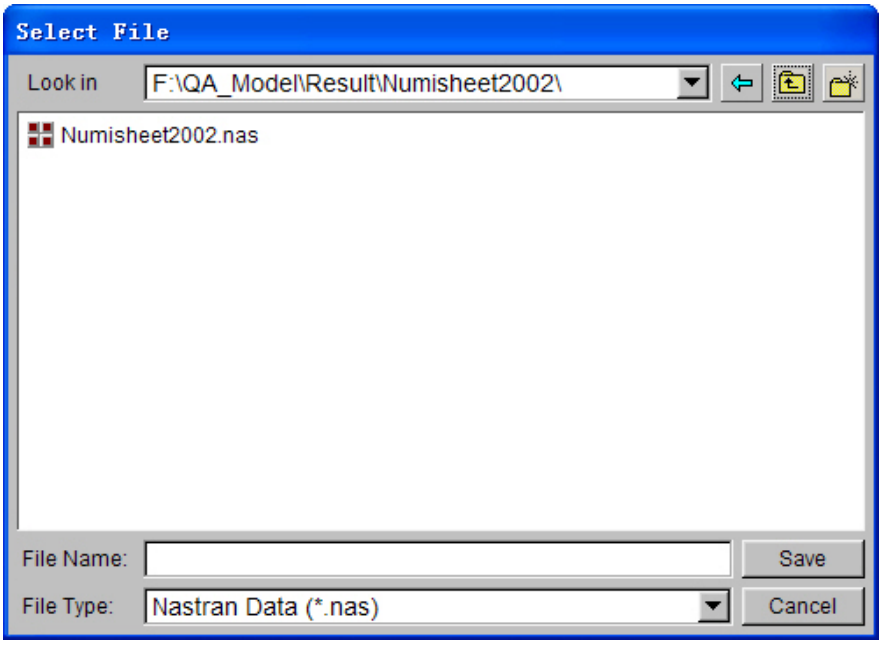


Figure 9.11 Export file dialog box

Usage: User can export the model turned on in the current database as the NASTRAN FILE. Give a file name, then click SAVE button to export the model in the selected file format.

Note: EXPORT function is disabled until a d3plot file or NASTRAN punch/output2 file is read in.

9.7.4 COPY TO CLIPBOARD

Description: This function allows the user to save the model displayed in the GRAPHICS DISPLAY window to the clipboard in the Windows environment. The image in the clipboard can be pasted to the document files.

9.7.5 PRINT

Description: The PRINT function prints the contents of the graphics display window to a printer or saves it to an image file.

This function allows the user to define the default settings for printing or saving image files. The options are shown in Figure 9.12.

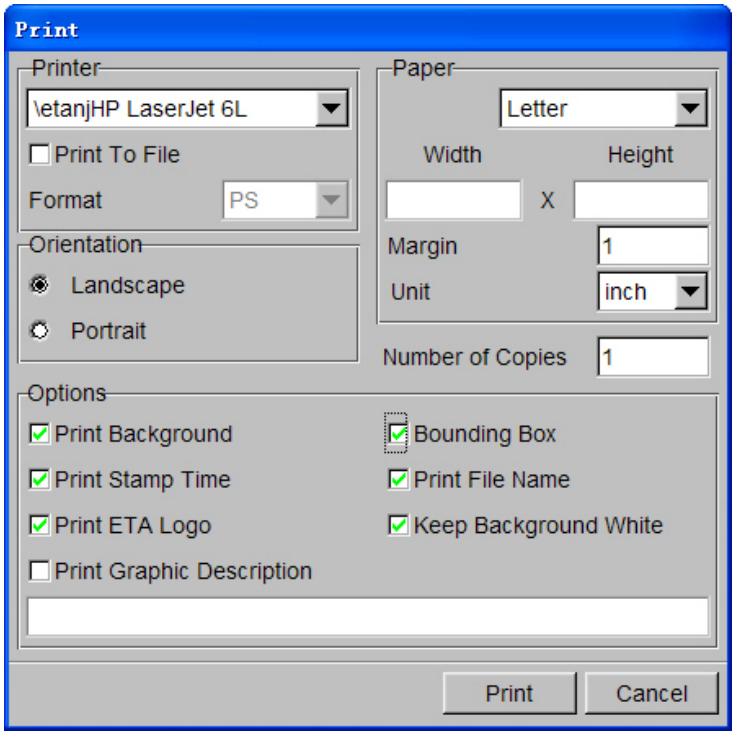


Figure 9.12 Print Window

9.7.5.1 PRINTER

Description: The user can select a printer or select a format to save to a file.

Usage:

a) Enter the printer name in the field or select from the drop down list and click PRINT to print the image.

b) If PRINT TO FILE option is selected, the user can choose a format by selecting the options in the drop down list in FORMAT window. DYNAFORM supports Postscript (PS), Encapsulated Postscript (EPS), GIF and JPEG file formats.

Note: When PRINT TO FILE is selected, the program will prompt the user to enter a filename to save the current display to a file of the specified format.

9.7.5.2 PAPER

Description: The user can specify the paper size and/or margin for the image print out.

Usage:

c) Select the button at the top left to choose a paper size.

Note: The supported sizes are: LETTER - 8.5x11 inches; A4 - 8.26x11.69; and B5 - 7.17x10.13. Users can also specify a paper size for a specific paper by entering size in the width and height field.

d) Enter a number in the field next to MARGIN to define the margin of the shorter edge of the paper. The program automatically determines the margin of the longer edge in order to maintain the original aspect ratio of the picture. This

feature can also be used to scale the picture.

e) Select the drop down option next to UNIT to choose the unit (inch or mm) used for paper size and margin.

9.7.5.3 ORIENTATION

Description: This function defines the image orientation as landscape or portrait on the printed copy.

9.7.5.4 OPTION

Description: These options are for defining printer output and layout on the paper.

Usage:

PRINT BACKGROUND
If this function is toggled on, the background color of the screen will be included in the print out. If this function is toggled off, there will be no background color on the print out.

BOUNDING BOX
This function draws a line frame around the picture's border.

PRINT TIME STAMP
This function prints the current time at the lower right corner of the picture.

PRINT FILE NAME
This function prints the file name at the lower left corner of the picture.

PRINT ETA LOGO
This function prints eta/VPG at the lower right corner of the picture.

PRINT GRAPHIC DESCRIPTION
If the function is toggled on, the field below the toggle switch is enabled to use. User can enter a string of characters to describe the current image to be printed on the print out.

9.7.5.5 NUMBER OF COPIES

Description: This function allows the user to print multiple copies to the printer. It has no effect on the PRINT TO FILE option.

9.7.5.6 PRINT

Description: This function will send the model to a selected printer or prompt the user to enter a file name to save the file.

9.7.5.7 CANCEL

Description: This function allows user to exit the function, and reject any selections made.

9.7.6 PRINT TO FILE

Description: This function prints the result to a file.

9.7.7 QUIT (ALT+Q)

Description: Selection of this option ends the current eta/Post session and returns back to eta/VPG pre processing environment. If the user is executing eta/Post from a stand-alone mode, this option terminates the eta/Post software.

9.8 EDIT MENU

The functions in the Edit menu allow the user to modify the setting of the model display. Figure 9.13 shows the drop down list of the Edit functions.

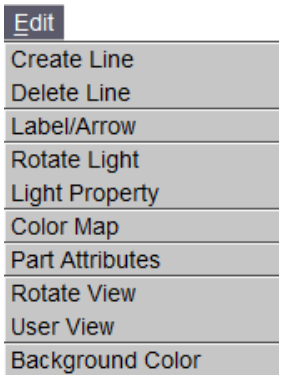


Figure 9.13 Edit menu

A detailed description of each function is given in the following sections.

9.8.1 CREATE LINE

Description: This function enables the user to create a line by selecting a set of nodes. In eta/Post a line is formed by a sequence of points. The program displays the line by a set of straight-line segments between adjacent points in sequence. The generated lines are included in a new part. The program displays a CONTROL OPTION window as shown as Figure 9.14.

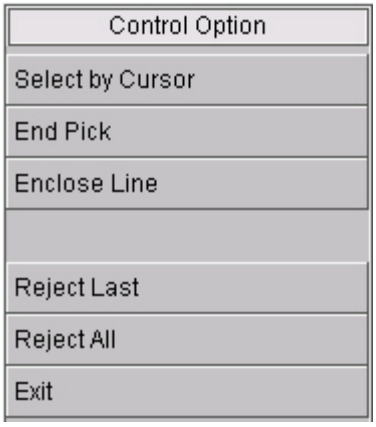


Figure 9.14 Create Line Control Option window

Usage:

- **SELECT BY CURSOR**
  Select the location of the node, a point will be created.

- **END PICK**
  Click this button to create a line by a set of straight-line segments between adjacent points in sequence.

- **ENCLOSE LINE**
  The created line is closed by a straight-line segment between the first selected and

the last selected point.

- **REJECT LAST**
  The last selected node is rejected.

- **REJECT ALL**
  All selected nodes are rejected.

- **EXIT**
  Exit the function.

9.8.2 DELETE LINE

Description: This function enables the user to delete the selected line(s).

9.8.3 LABEL/ARROW

Description: LABEL allows the user to enter a title or text label at any location in the graphic display window. ARROW allows the user to draw arrows at any location in the display window. The program displays a control window as shown in Figure 9.15.

Usage:

- **DISPLAY LABEL/ARROW**
  Controls the Label/Arrow display in the graphic display window. The default setting is toggled on.

- **ARROW**
  This function enables the user to select two points by cursor to draw an arrow in the graphic display window. The name of the arrow will be listed in the Label/Arrow control window. The first location is the tail of the arrow and the second location is the head of the arrow.

- **TEXT**
  This function allows the user to add a text label in the graphic display area. Enter a string of characters in the field above the TEXT button. Press the TEXT button then click a location on the screen. The program will add the text label at the clicked location.

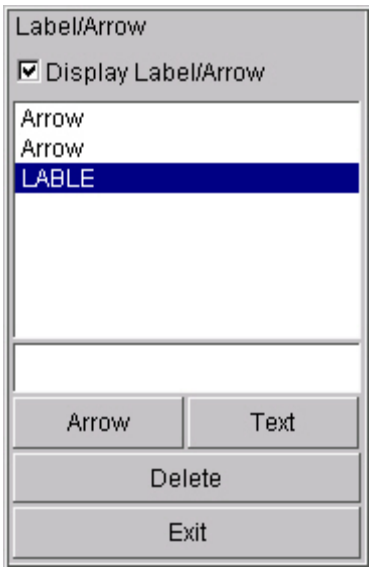


Figure 9.15 Label and Arrow Control Option Window

- **DELETE**
  Delete an arrow or a text label. The program will highlight the arrow or text label on the screen as the user selects an arrow or text label in the list. Click the DELETE button to delete the highlighted arrow or text label. The user may combine the Shift or Ctrl key and mouse click for multiple selections.

- **EXIT**
  Exit the function.

9.8.4 ROTATE LIGHT

Description: eta/Post uses two light sources directed from the specific locations from the model. This function allows the user to rotate the light sources along the screen X Y axes. The function only works when the SHADE option is turned on. The program displays the light source 1 and 2 and their lighting directions when the function starts.

Usage: The user uses the mouse to move the light sources on the screen. The lighting effect is updated as the user moves the mouse. Press the left mouse button to exit the function. The user may activate this function by pressing the SHIFT and LEFT mouse button simultaneously. Release the mouse button to exit the function.

9.8.5 LIGHT PROPERTY

Description: The light property allows the user to adjust the brightness and shininess of the part by adjusting the setting of Ambient, Diffuse and Specular light. The LIGHT PROPERTY only is operational when the SHADE option is turned on. The user may click and drag the slider in each light property to adjust the light property. Slide to the right results in more brightness or reflection of the display.

Usage:

- **AMBIENT**
  Ambient light is a uniform light source coming from all directions to the part.
- **DIFFUSE**
  Diffuse light is a parallel light source coming from the light source direction. This light is reflected evenly from the part surface.
- **SPECULAR**
  Specular light is similar to the diffuse light except the light is reflected sharply in a particular direction.
- **RESET LIGHT**
  Reset the light property to the default setting.

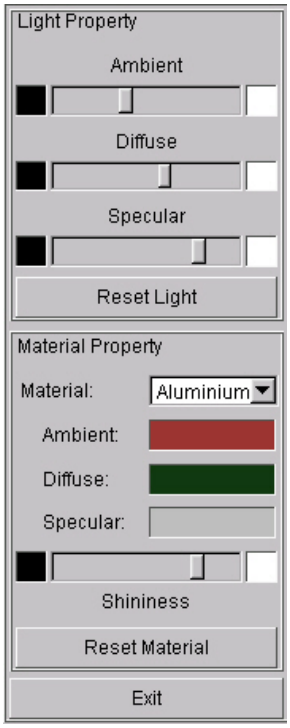


Figure 9.16 Light Property Control Option Window

9.8.6 MATERIAL PROPERTY

Description: eta/Post allows the user to modify different material properties for rendering the part. The material property only takes effect in the SHADE mode with GRAY COLOR option.

Usage:

- **MATERIAL**
  User may select the material from the Material drop down list. The available materials are: aluminum, steel, copper, iron, silver, bronze and rubber. User may assign a material to selected parts by using the PART ATTRIBUTE function.
- **MATERIAL COLOR**
  The program provides adjustment for

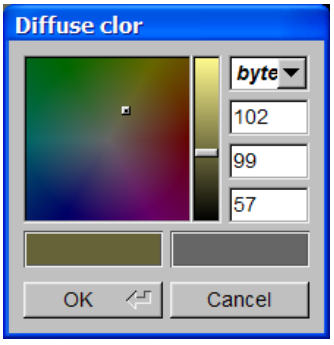


Figure 9.17 Light Adjustment Control Window

AMBIENT, DIFFUSE and SPECULAR color by clicking the color box next to the property type. The program displays a control window as shown in Figure 9.17

The user may click and drag the marker in the color map window to change the color of the selected material. The program updates the model display with the new material color in the graphic display window. It also shows the new material color in the color box on the low left side of the control window. The user can compare the new material color with the original color on the right. The user may also click and drag the slider in the vertical bar to adjust the brightness of the selected material. The RBG values of the material color are shown in the column on the right side of the control window. The user may change these values to change the material color. The user may choose any of the four scales to show the RGB values: Byte, RGB, Hex or HSV. When the desirable color is obtained, click OK to accept the color and exit the control window. Otherwise, click CANCEL to reject the color and exit the control window.

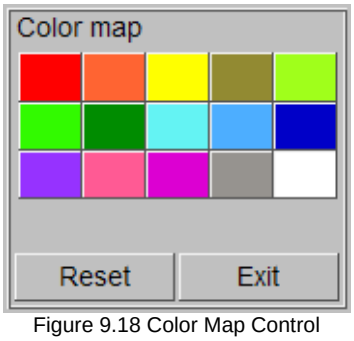


Figure 9.18 Color Map Control

- **SHININESS**
  The user may click and drag the slider to adjust the shininess of the selected material.

- **RESET MATERIAL**
  Resets the material color to the default setting.

- **EXIT**
  Close the light property control window and exit the function.

9.8.7 COLOR MAP

Description: Eta-Post uses 14 different colors to distinguish parts in the model. This function enables the user to modify any part color from default setting. The program displays the COLOR MAP control window as shown in Figure 9.18

Usage: The user may select any color to modify by clicking on a color block in the color palette. The program will display a change part color control window as shown in Figure 9.19

The procedure of changing Part Color is same as in Change Material Color that has been described in Section 4.6.

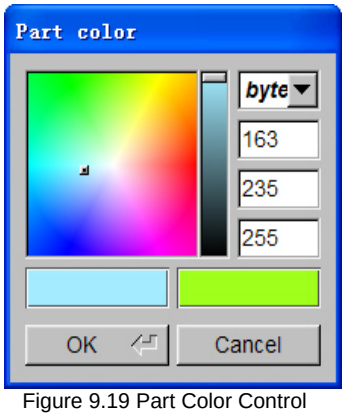


Figure 9.19 Part Color Control

9.8.8 PART ATTRIBUTES

Description: The functions in PART ATTRIBUTES control window allow the user to customize the display characteristics for selected parts in the model. There are fives options. The default PART ATTRIBUTES control window is shown in Figure 9.20.

Usage:

- **PART COLOR**
  The user may change part color from the color list. Click on the PART COLOR option and select the parts from the part list window or click the part from the screen.

The program will mark the part name with an asterisk (*) and highlight the parts on the screen. Select a color from the color table to change the part color. Then click APPLY below the control window. The program will change the color of the selected part on the screen and the part name in the window. Please see the Figure 9.20.

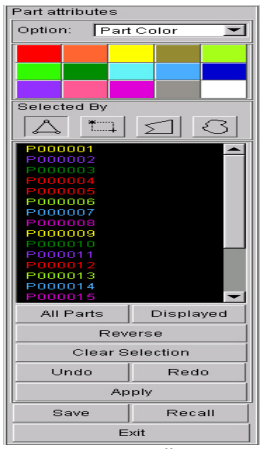


Figure 9.20 Part Attributes control window (Part Color)

- **PART MATERIAL**
  This option allows the user to assign material property to the selected parts when the GRAY SHADE option is turned on. The available materials are: aluminum, steel, copper, iron, Gold, silver, bronze and rubber. Click on the MATERIAL button and select a material type from the draw down list. Then the user can select the parts from the part list window or click the parts from the screen. The program will mark the part name with an asterisk (*) and highlight the part on the screen. Clicks APPLY and the selected parts will be endued with the specified material. The changes of part's material can be show only when the SHADE and GRAY option are turned on. Please see the Figure 9.21.

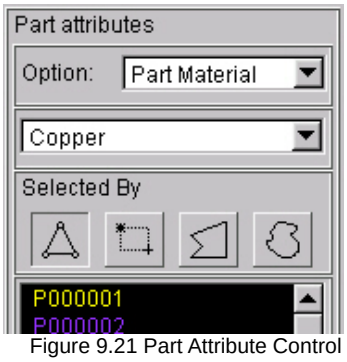


Figure 9.21 Part Attribute Control Window (Part Material)

- **TRANSPARENCY**
  This option allows the user to make the selected parts in transparent when the SHADE option is turned on. Click on the TRANSPARENCY button and select the parts from the part list window or click the parts from the screen. The program will mark the part name with an asterisk (*) and highlight the part on the screen. Clicks APPLY and the transparency slider will be active. The user may use the transparency slider to adjust the degree of transparency. Slide to the left will make the selected parts more transparent. The program will show the selected parts transparent when the SHADE option is turned on. Please see the Figure 9.22.

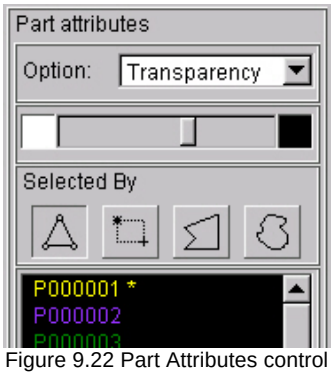


Figure 9.22 Part Attributes control window (Transparency)

- **WIREFRAME**
  This option allows the user to display the selected parts in wire frame when the SHADE option is turned on. Click on the WIREFRAME button and select the parts from the part list window or click the parts from the screen. The program will mark the part name with an asterisk (*) and highlight the part on the screen. The program will not shade the selected parts when the SHADE option is turned on. Please see the Figure 9.23.

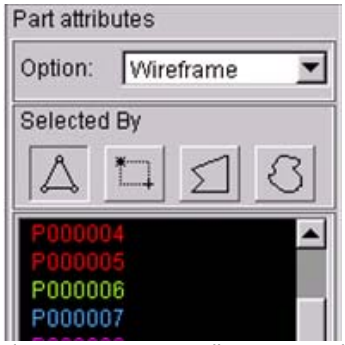


Figure 9.23 Part Attributes control window (Wire Frame)

- **NO CONTOUR**
  The user may choose not to show contour result in select parts. Click on the NO CONTOUR option and

select the parts from the part list window or click the part from the screen. The program will mark the part name with an asterisk (*) and highlight the parts on the screen. Clicks APPLY and the program will not display contour of the selected part during CONTOUR ANIMATION. Instead, the selects parts will be plotted in gray color. Please see the Figure 9.24

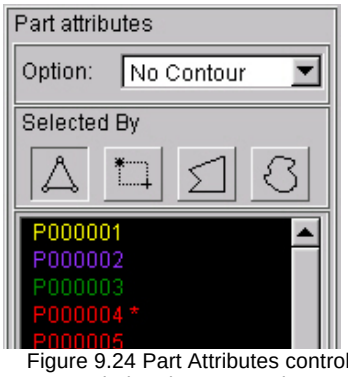


Figure 9.24 Part Attributes control window (No Contour)

There are some functional buttons on the Part Attributes window for the user to easily select groups of parts.

ALL PARTS Enable the user to select all the parts on the current database.

DISPLAYED Enables the user to select all the parts that displaying on the current window.

REVERSE Enables the user to reverse the selected parts. All the selected parts will be unselected and all the unselected parts will be selected.

CLEAR SELECTION Turn all the selected parts to unselected.

UNDO Enables the user to cancel the last selected operation before pressing the Apply button.

REDO Enables the user to cancel the UNDO operation.

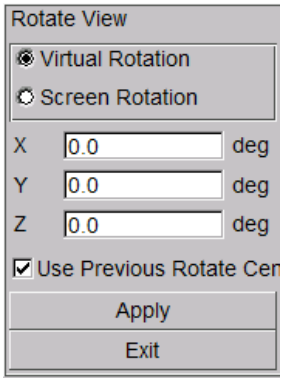
APPLY Enables the user to execute the selected operation.

SAVE Enables the user to save the current part attributes in file. A Select file dialog box will pop up and prompt the user to specify a name after clicking this button.

RECALL Enables the user to load the previously saved parts attribute file and set as the current parts attribute.

9.8.9 ROTATION VIEW

Description: This function is used to finish virtual rotation or screen rotation, for virtual rotation, model rotates about global axis, for screen rotation, model rotates about screen axis.



9.8.10 USER DEFINED VIEW

Description: This function is used to save the current viewing orientation and recall the previously saved view. User may save up to 10 views in a session. The program displays a control window as shown in Figure 9.25

Figure 9.25 User View control window

Usage:

- **SAVE VIEW**
  Saves the current viewing orientation. The program assigns a default view name as shown in Figure 9.26. The User may enter any view name in the VIEW NAME field. Click OK to save the current viewing orientation, or click CANCEL to abort the save operation.

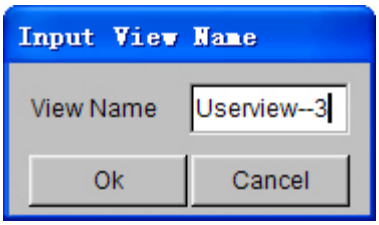


Figure 9.26 View Name control window

- **RECALL VIEW**
  Recalls the previously saved viewing orientation. Click a VIEW NAME in the view list and then click RECALL. The program will display the model according to the viewing orientation saved under the selected VIEW NAME.

- **DELETE**
  Deletes the previously saved viewing orientation. Click a VIEW NAME in the view list and then click DELETE to delete the saved view.

- **EXIT**
  Closes the SAVE VIEW control window and exits the function.

9.8.11 BACKGROUND COLOR

Description: This function allows the user to select a color from color palette as the background color in the graphic display window. Once the function is selected, the program displays a color palette as shown in Figure 9.27.

Figure 9.27 Background Color Palette

Usage: The user may select any color in the palette as the background color. The program will change the background color immediately and exit the function as the color is selected. Press the ESC key to exit the function without selecting any color.

9.9 TOOL MENU

The functions in this menu are shown in Figure 9.28. The user can define section cut, mirror result, define active window, trace node and define node curve.

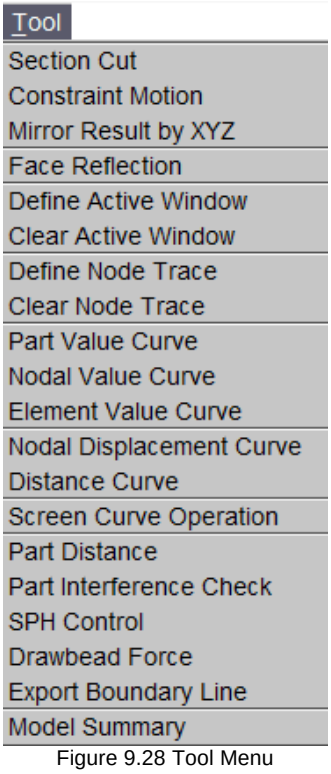


Figure 9.28 Tool Menu

A detailed description of each function and corresponding submenu is given in the following section.

9.9.1 SECTION CUT

Description: This function displays the section cut of a finite element model with a defined plane intersecting the edges of elements. The functions in SECTION CUT control window are shown in Figure 9.29. DEFINE CUT PLANE is the only option available when the SECTION CUT functions starts.

Figure 9.29 Section Cut Operation

Usage:

- DEFINE CUT PLANE
  The section plane is the U-V plane of a user defined local coordinate system. The local coordinate system may be defined by selecting one, two or three nodes from the model.

  The program displays a Control Option window as shown in Figure 9.30. The program also changes the displayed model to gray color.

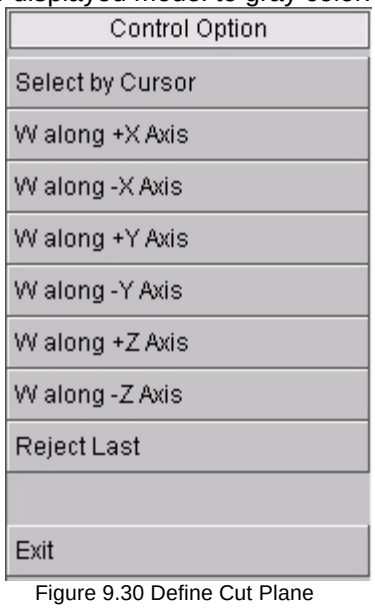


Figure 9.30 Define Cut Plane

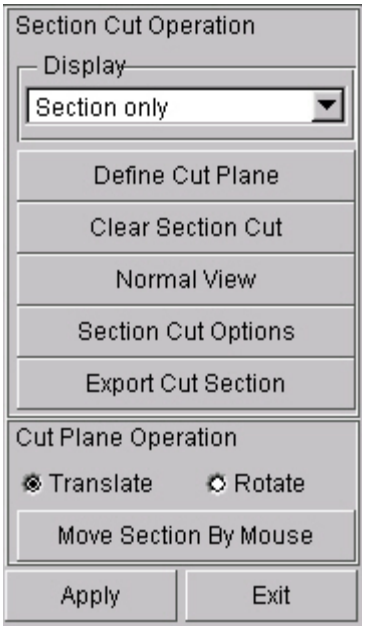


Figure 9.31 Section Cut Operation

ONE POINT

Select a node by cursor to define the origin of the LCS, then select a global direction as the local W axis and click EXIT. The global direction may be either + or – X (Y or Z) axis as listed in Figure 9.31. The local coordinate system will be defined at the first node with the local W axes follow the selected direction and U, V axes parallel to the other two global axes.

TWO POINTS

Select the first node to define the origin of the LCS and select second node to define the local W-axis and click EXIT to define the LCS. The local coordinate system will be defined at the first node with local w-axis aligned with the vector connecting the first and the second node.

THREE POINTS

Select the first node to define the origin of the LCS, select the second node to define the local U axis and the third node to define the U-V plane. The local V and W axis will be computed according to the right hand rule.

Once the local system is defined and accepted, the program will display the section lines in part color that represent the section cut of the model in the u-v plane. The program will activate other options as shown in Figure 9.32 for the user to change or operate the section cut.

- CLEAR SECTION CUT
  This function enables the user to erase the section cut from the display window.
- NORMAL VIEW
  This function enables the user to view from the normal of the cut plane.
- SECTION CUT OPTIONS
  This function controls the export and display of the section cut.

  CUT PLANE

The default number of cut planes is 20 if ALL PLANE option is selected. This means there will be 20 cut planes equally spaced in the local W direction. If the CUR.PLANE is toggled on, the current section cut will be exported when select the function EXPORT SECTION CUT. If the ALL PLANE option is selected, all 20 section cuts will be exported when select the function EXPORT SECTION CUT.

MAX RADIUS
Set the maximum arc radius that can be displayed.

MIN RADIUS
Set the minimum arc radius that can be displayed.

RADIUS DEV.
Set the tolerance between two adjacent arcs to merge.

MIN CHORD
Set the minimum chord length that can be displayed.

SECTION PLUS CURVE
This function is only enabled for the section cut of contour mode. Toggle on the check box, then press APPLY button, the curve window will display. Please see the Figure 9.32. In the curve window, the abscissa is the arc length and the ordinate is the corresponding contour value.

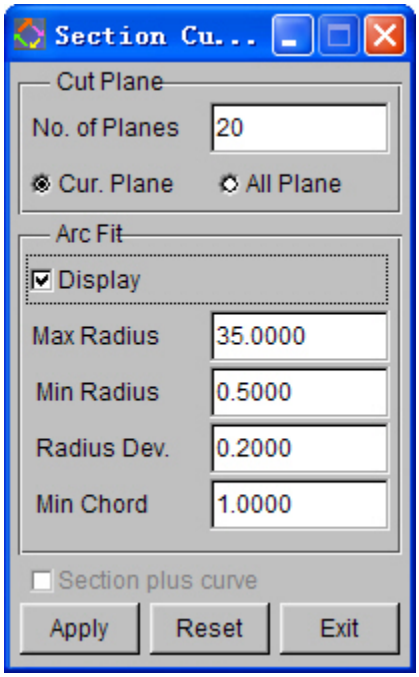


Figure 9.32 Section Cut Operations

EXPORT SECTION CUT
This function allows the user to export the section cut as the PLOTEL elements to a NASTRAN file. The program displays a Select File dialog window as shown in Figure 9.33.

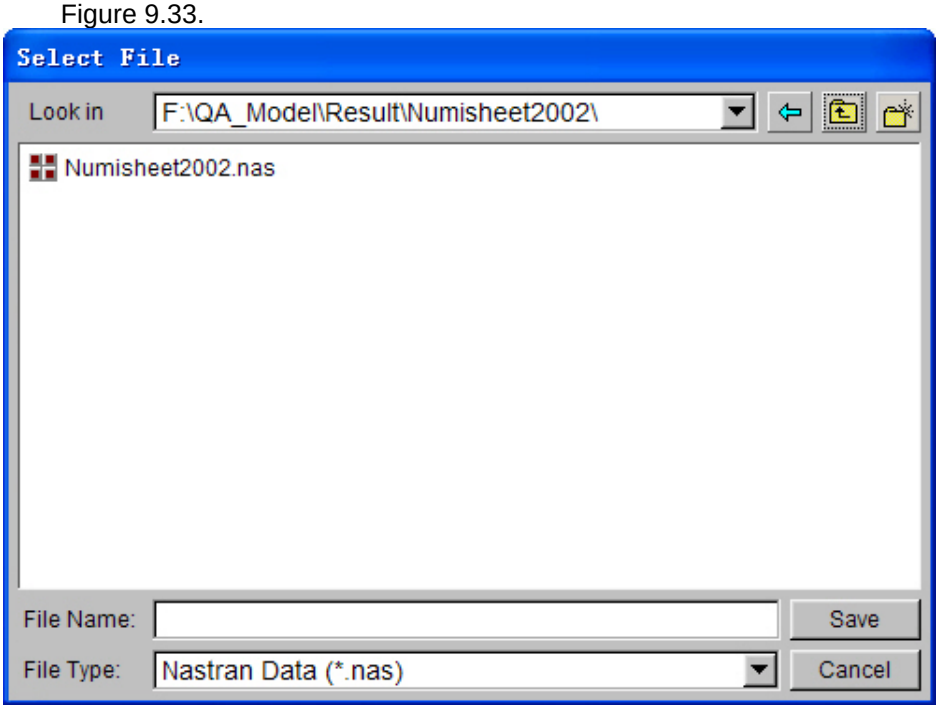


Figure 9.33 Export Nastran File

MOVE SECTION BY MOUSE

This function allows the user to move the section cut location by using the mouse. The movement may be translation along the local W axis or rotation about the local w axis.

TRANSLATE

First select the MOVE SECTION BY MOUSE button. eta/Post allows the user to select a node on the current section as the reference node. The program draws a line along the local W axis as the directional line. Drag the mouse along the directional line to obtain the new section cut nearest the cursor. Click the left mouse button to exit the function.

ROTATE

First select MOVE SECTION BY MOUSE button. eta/Post allows the user to select two nodes on the current section as the U-Axis. Exit the Control Option to accept the U-Axis, at this time the user is allowed to drag mouse, the section cut will rotate along the U-Axis.

DISPLAY OPTIONS

SECTION ONLY

Only the section line will be displayed after exit from the section cut function.

WHOLE MODEL

Whole model (in grey) will be displayed with section line after exit from the section cut function.

SECTION CUT PORTION

This function allows user to display a portion of the model on the positive or negative side of the local W-axis after exit from the section cut function. The program will display only the selected portion of the model on the screen.

Figure 9.34 shows a typical display of SECTION ONLY option.

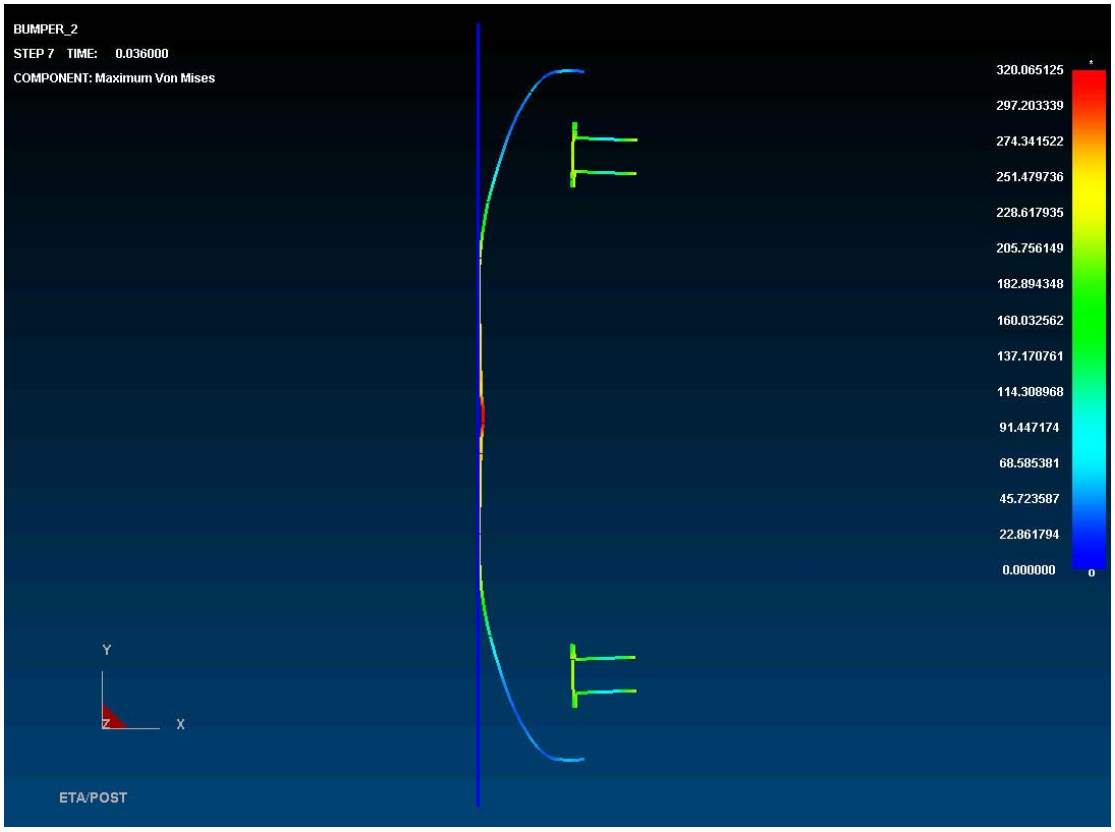


Figure 9.34 An Example for SECTION CUT display

9.9.2 CONSTRAIN MOTION

Description: This function enables the user to define a reference point for animation. The user may select any node in the model as the reference point by the function **SELECT REFERENCE NODE**. The user may also select any or all translation degree of freedom (X, Y and Z) to constrain the motion. Figure 9.35 shows the dialog window of the Constraint Motion.

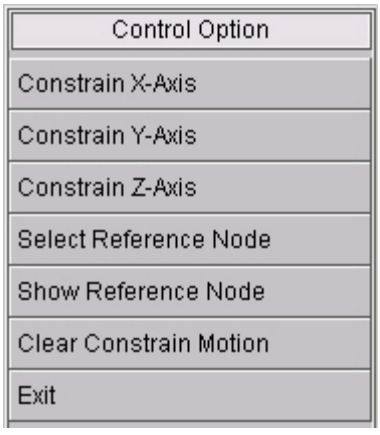


Figure 9.35 Constraint Motion Dialog Window

Disregarding the displacement of the node, it will be stationary in the constrained direction during the animation. The rest of the model will be displaced relative to the reference point in

the un-constrained direction(s).

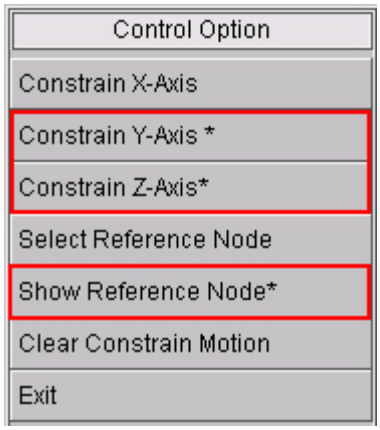


Figure 9.36 Constraint Motion Dialog Window

Note: When Axis is selected as the constraint axis, the function will be labeled with asterisk. When toggle on Show Reference Node, the function will be labeled with asterisk, and when animate, the reference node will be labeled with highlighted circle. Please see the Figure 9.36.

9.9.3 MIRROR RESULT BY XYZ

Description: This function allows the user to select a mirror plane to mirror the analysis result. There are three mirror planes: XY PLANE, YZ PLANE and ZX PLANE as shown in Figure 9.37.

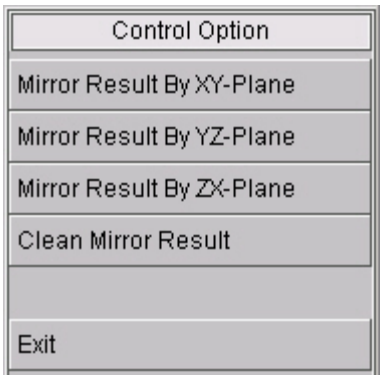


Figure 9.37 Mirror Result Control Option Window

Usage: The following example is to mirror a quarter of the model and result as shown in Figure 9.38. When selecting MIRROR RESULT BY XZ PLANE, the program mirrors the model and result as shown in Figure 9.39.

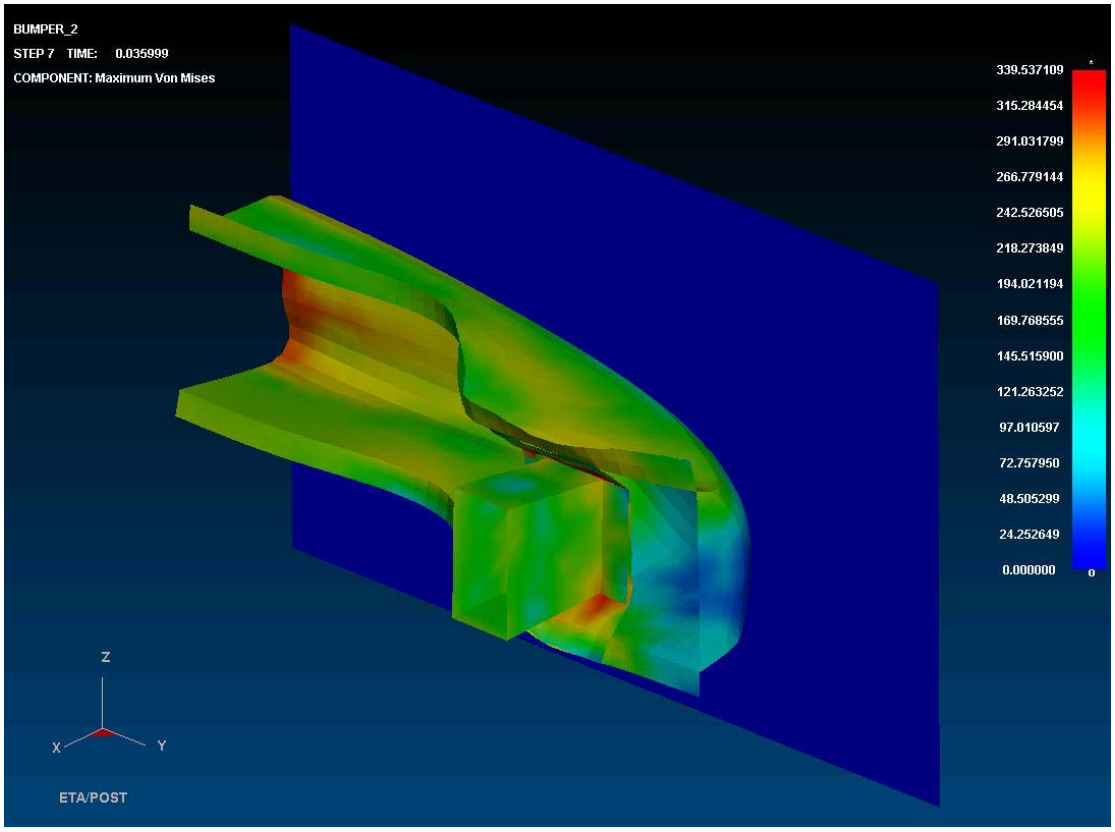


Figure 9.38 Example of Mirror Function Input

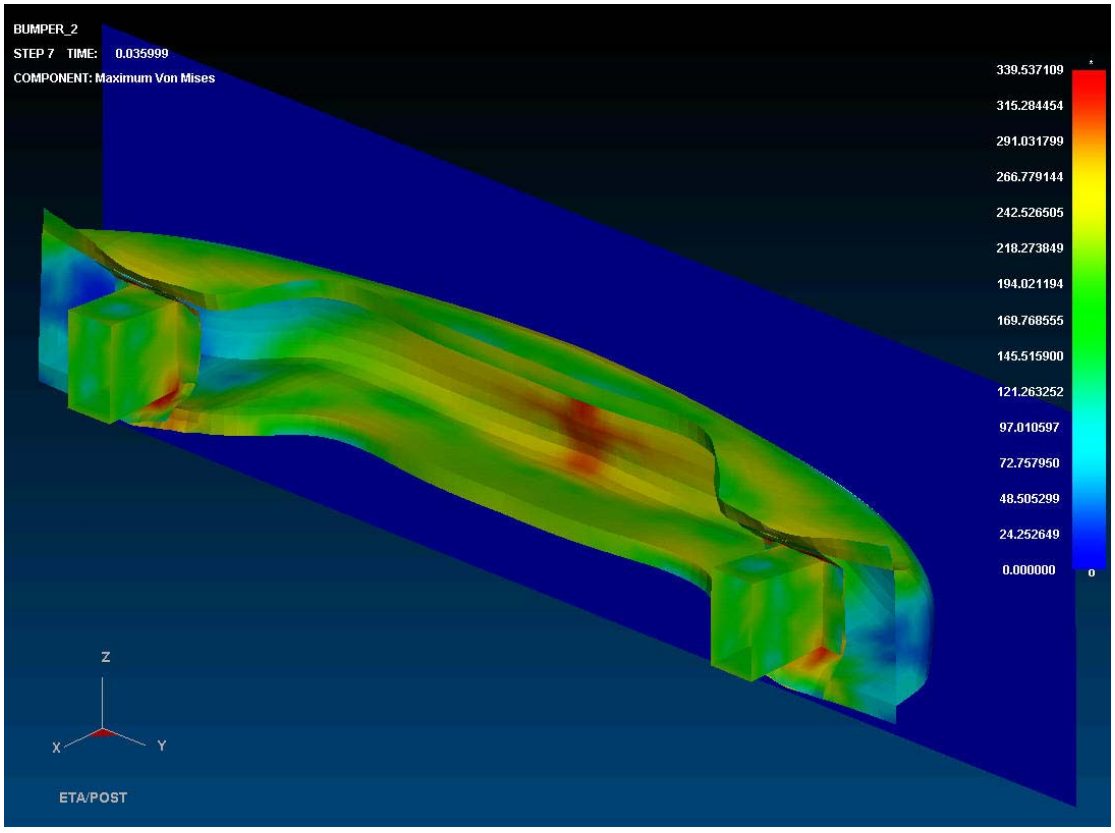


Figure 9.39 Mirror Results by XZ Plane

Press the CLEAN MIRROR RESULT on the Control Option Window, the mirror result is missing and the model is displayed as the original status.

9.9.4 DEFINE ACTIVE WINDOW

Description: This function allows the user to isolate a portion of the displayed model for more detailed viewing or editing. The user defines the desired area. eta/Post displays the elements with the analysis result as active window. Other objects on the screen will be masked and inactive.

Usage: The user can define the Active Window by Cursor, Drag Window, Polygon Freehand and Displayed. See the Control Option shown in Figure 9.40.

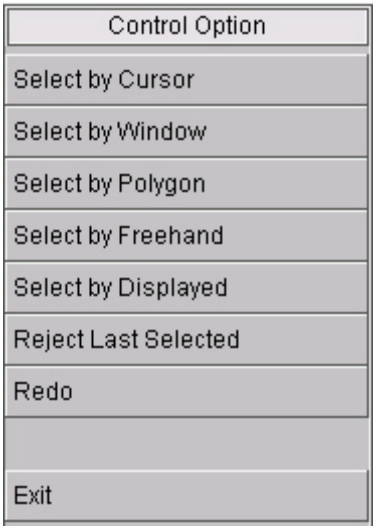


Figure 9.40 Define Active Window Control Window

Click CLEAR ACTIVE WINDOW from the TOOL Pull-down menu to remove the defined active window. The program will display the whole model as the original status.

9.9.5 DEFINE NODE TRACE

Description: NODE TRACING allows the user to select a set of nodes to be traced during animation. A trace is a line or track that identifies the path that a node takes during its movement from one time step to another. An example of the Node Trace is show in Figure 9.41.

Usage:

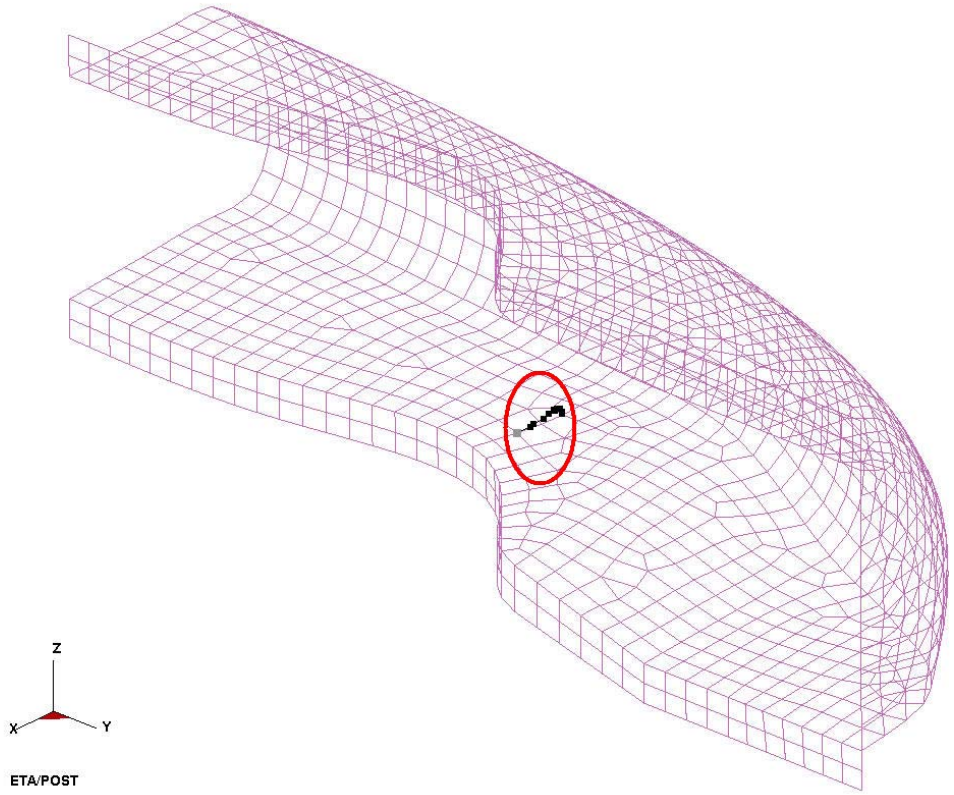


Figure 9.41. The Node Trace

Click the CLEAR NODE TRACE from the TOOL Pull-down menu to remove all of the currently defined traces.

9.9.6 PART VALUE CURVE

Description: This function allows the user to plot the maximum and minimum node results in a time history graph of the selected part. The function only works during Contour Animation with ELEMENT RESULT options selected.

9.9.7 NODAL VALUE CURVE

Description: This function allows the user to plot the result in a time history graph of the selected nodes. The function only works during Contour Animation. The user should start a contour animation of a selected result component before using this function. The program displays a dialog window as shown in Figure 9.42.

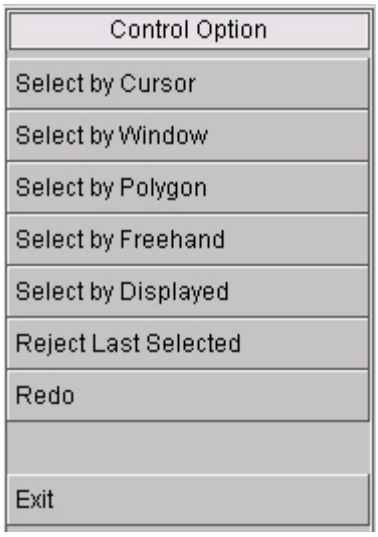


Figure 9.42 Select Nodes Control Option

The procedure of selecting nodes is common with other functions described in previous sections. After selecting desired nodes, exit the function, the node curve result is shown as in Figure 9.43.

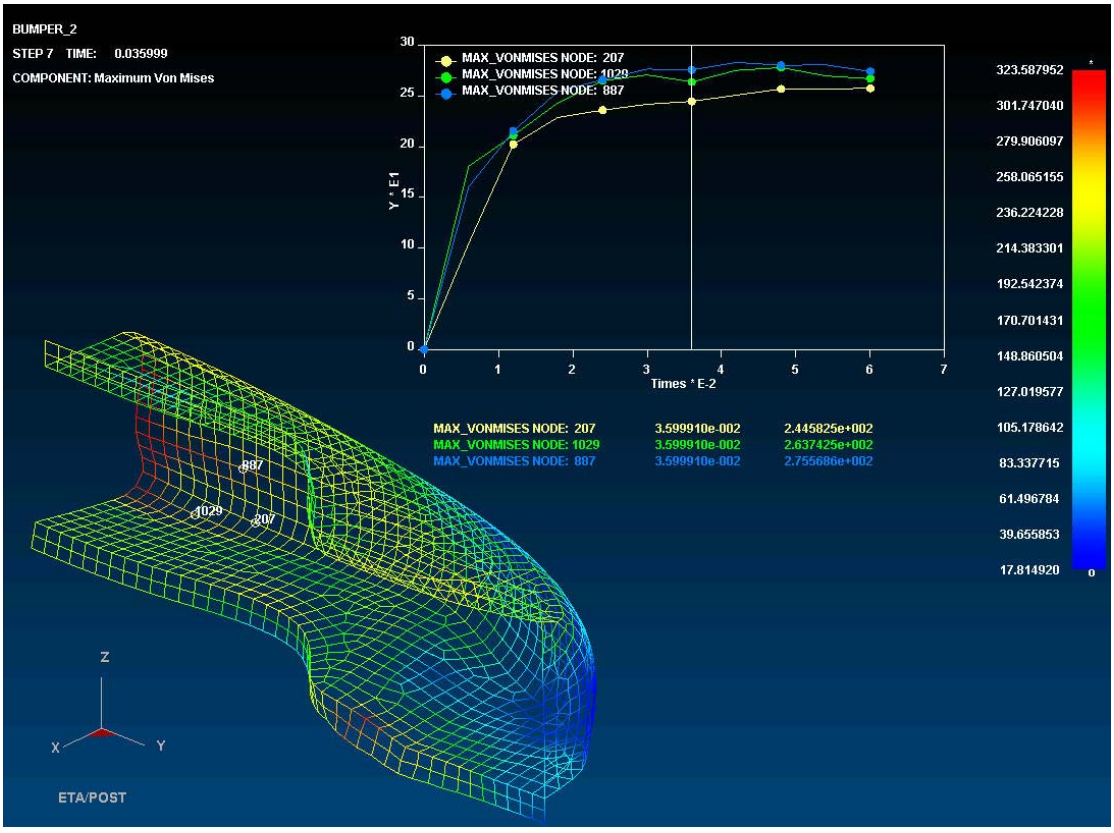


Figure 9.43 Node Curve Result

9.9.8 ELEMENT VALUE CURVE

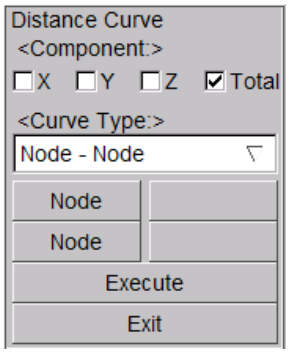
Description: This function allows the user to plot the result in a time history graph of the selected elements. The function only works during Contour Animation with ELEMENT RESULT options selected.

9.9.9 NODAL DISPLACEMENT CURVE

Description: This function allows the user to plot the displacement result in a time history graph of the selected node. The function only works during Contour Animation with ELEMENT RESULT options selected.

9.9.10 DISTANCE CURVE

Description: This function allows the user to plot the distance result in a time history graph. This distance can be node to node, to element, to part and to plane. The function only works during Contour Animation with ELEMENT RESULT options selected.



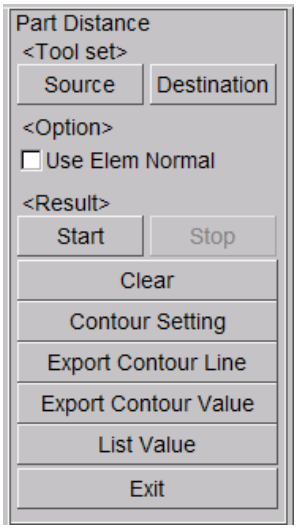
Usage: The user first selects a node, then select another node, an element, a plane or a part, last click EXECUTE, the distance will be plotted in the graph.

9.9.11 SCREEN CURVE OPERATION

Description: This function allows the user to do some operations to curves last plotted. User can on/off, hide, show, delete and export curves in the list.

9.9.12 PART DISTANCE

Description: This function will show the distance of source part from destination part.

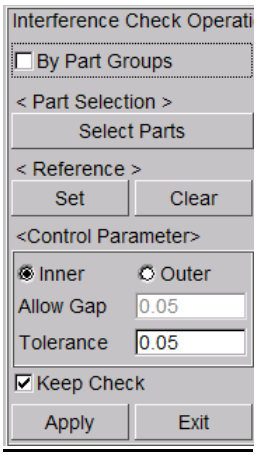


Usage: select a source part and a destination part, and then click button START, the distance of source part from destination part will be shown as contour. After that, user can clear, export and list value of this contour.

9.9.13 PART INTERFACE CHECK

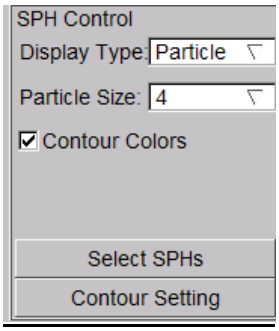
Description: This function allows the user check interference between parts. Interference elements will be shown highlighted.

Usage: After selecting parts, click button APPLY, the interference will be shown highlighted. User can define search tolerance for check result.



9.9.14 SPH CONTROL

Description: This function allows the user to define options of showing SPH results.



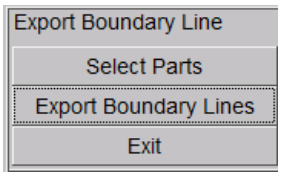
Usage: Select particle, ribbon or vector as the SPH display type, and then define size of particle, ribbon or vector. User can select all or a portion of particles to show result.

9.9.15 DRAWBEAD FORCE

Description: This function allows the user to show drawbead force for metal forming.

9.9.16 EXPORT BOUNDARY LINE

Description: This function allows the user export boundary lines of parts



Usage:

1. User selects one or more parts, and boundary lines of selected parts will be shown highlighted.
2. Export the highlighted boundary lines, saved as format of *.lin

9.9.17 MODEL SUMMARY

Description: This function enables the user to display the statistics regarding elements and nodes in the model. Figure 9.44 shows a sample model summary.

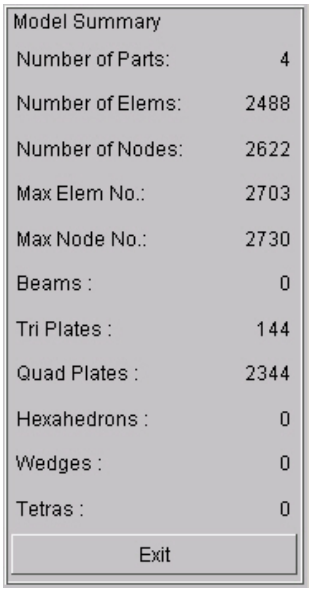


Figure 9.44 Model Summary

9.10 OPTION MENU

The functions in this menu are shown in Figure 9.45. The user can customize the appearance in the graphic display window.

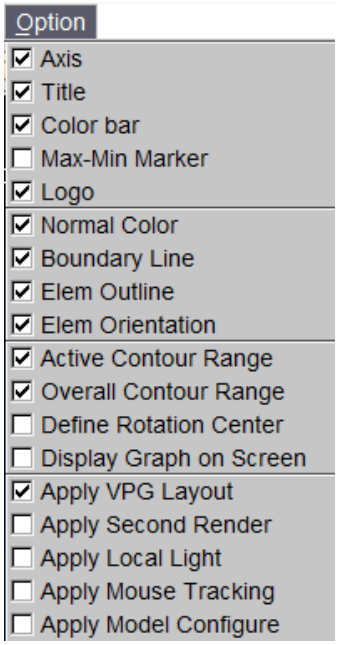


Figure 9.45 Tool Menu

A detailed description of each option is given in the following section.

9.10.1 AXIS (toggle)

Description: The XYZ coordinate system is displayed in the lower left corner of the display window. This option allows the user to toggle the axes on/off.

9.10.2 TITLE (toggle)

Description: The title of the result file is displayed in the upper left corner of the display window. This option allows the user to toggle the title on/off.

9.10.3 COLORBAR

Description: This function shows color bar when animate results

9.10.4 MIN-MAX MARKER

Description: This function shows the minimum and maximum marker at relative value position

9.10.5 LOGO (toggle)

Description: This function toggles the logo (ETA/POST) at the lower left corner of screen on and off.

9.10.6 NORMAL COLOR (toggle)

Description: This option allows the user to display the part color on positive side and gray color on the negative side of the finite element model when PLATE NORMAL option is selected in the DISPLAY OPTIONS window. This option is essential for the user to visually check the part for reversed normal of the finite element mesh. Figure 9.46 shows a part displayed with NORMAL COLOR and PLATE NORMAL option.

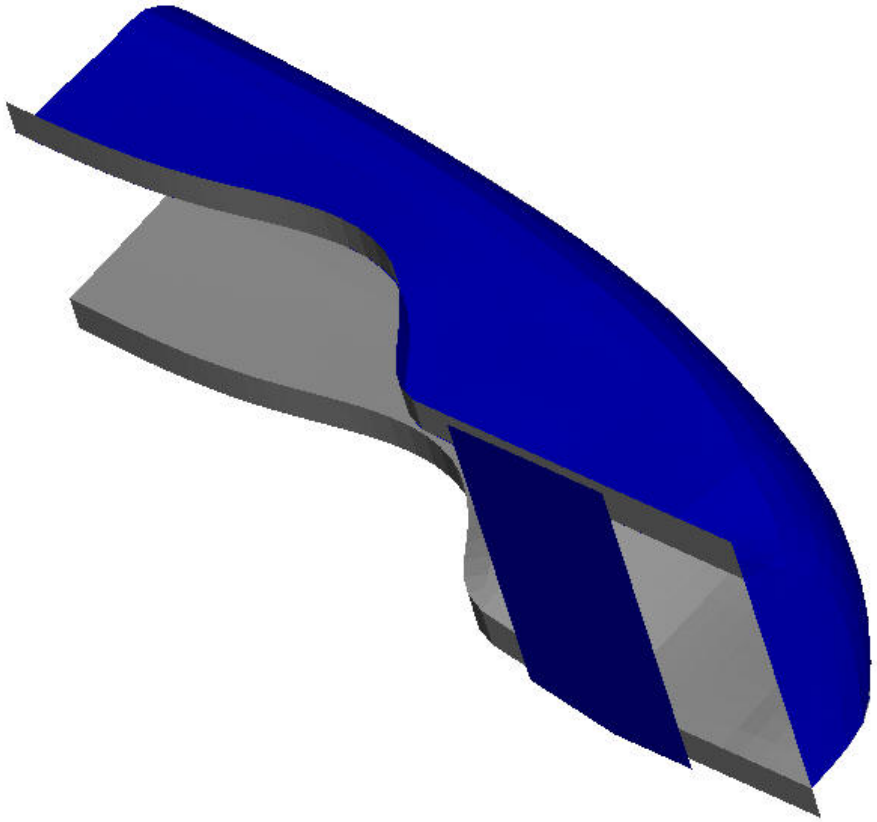


Figure 9.46 Typical display with Normal Color and Plate Normal option

9.10.7 BOUNDARY LINE (toggle)

Description: Boundary lines are shown black when this option is on.

9.10.8 ELEM OUTLINE (toggle)

Description: Element outlines are shown black when this option is on.

9.10.9 ELEMENT ORIENTATION (toggle)

Description: This option shows a vector from the first node to the second node of each plate element. It allows the user to visualize the direction of the element location U axis and local Z axis (according to the right hand rule). It is useful to check the orientation of the composite material. Figure 9.47 shows a typical display of a part with ELEMENT ORIENTATION option.

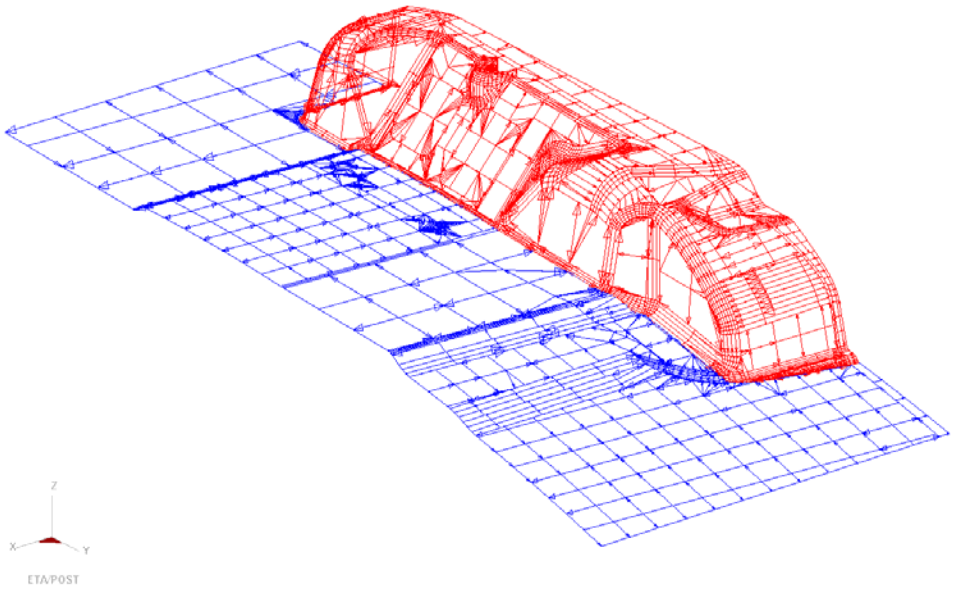


Figure 9.47 Typical display with Element Orientation option

9.10.10 ACTIVE CONTOUR RANGE

Description: This function will toggle on/off active contour range.

9.10.11 OVERALL CONTOUR RANGE

Description: This function will toggle on/off overall contour range.

9.10.12 DEFINE ROTATE CENTER

Description: This function will toggle on/off rotate center

9.10.13 DISPLAY GRAPH ON SCREEN (toggle)

Description: Graphs will directly be shown in display window without creating a new window when this option is on.

9.10.14 APPLY VPG LAYOUT

Description: This function will toggle on/off initial status of eta/post for VPG or DYNAFORM.

9.10.15 APPLY SECOND RENDER

Description: This function will toggle on/off second render.

9.10.16 APPLY LOCAL LIGHT

Description: This function will toggle on/off local light.

9.10.17 APPLY MOUSE TRACKING

Description: This function will toggle on/off mouse tracking

9.10.18 APPLY MODEL CONFIGURATION

Description: Configurations will be written into configure file, and set as default parameters when start eta/post next time if this option is on.

9.11 POSTPROCESS

The functions in the POSTPROCESS menu allow the user to graphically display and manipulate analysis results. There are 4 functions shown in the post-processing tool bar as shown in Figure 9.48.

Figure 9.48 Post-process functional icons.

Once a function in the tool bar is selected, the POST-PROCESSING CONTROL WINDOW appears. Each function allows the user to graphically display and manipulate a specific result for stamping simulation. The user selects the desired PLOT STATE, TIME STEPS, FRAME RANGE and COMPONENTS to plot or animate the result.

Note: **To load LS-DYNA result files and to activate the post process menu, see FILE/OPEN in Chapter 3 FILE MANAGER.**

Note: **The user will not have access to have the control window when other control option windows are active.**

A detailed description of each function is given in the following sections.

UNDEFORM

This function shows the undeformed status of the model.

DEFORM

The functions in this menu animate the displacement of the model in real time and display displacement of an individual step. The options are shown in Figure 9.49.

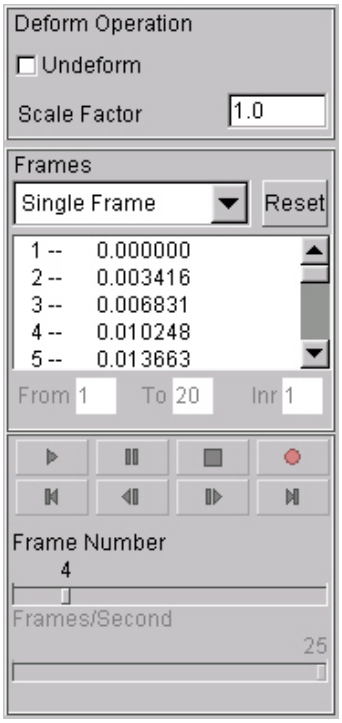


Figure 9.49 Deform control window

9.11.1 DEFORM OPERATIONS

- **SHOW UNDEFORM SHAPE**
  This function enables user to toggle the undeformed geometry of the model on/off. The deformed shape plot is displayed in its original color. The plot of the undeformed shape is displayed in white over the deformed shape.
- **SCALE FACTOR**
  This command adjusts the default scale factor of the deformed shape plot. The default is 1.
  It scales plot components to a user-defined value allowing the user to magnify or minimize their visual display. For example, if the plot state is set to deformation, the user could magnify the deformation by a factor of ten to see small deformation not readily visible during animation.

9.11.2 FRAME OPERATIONS

Description: This function allows user to select the desired frame(s) to plot (single frame) or animate (more than one frame) deformation shown as Figure 9.50. There are 6 options shown as Figure 9.51.

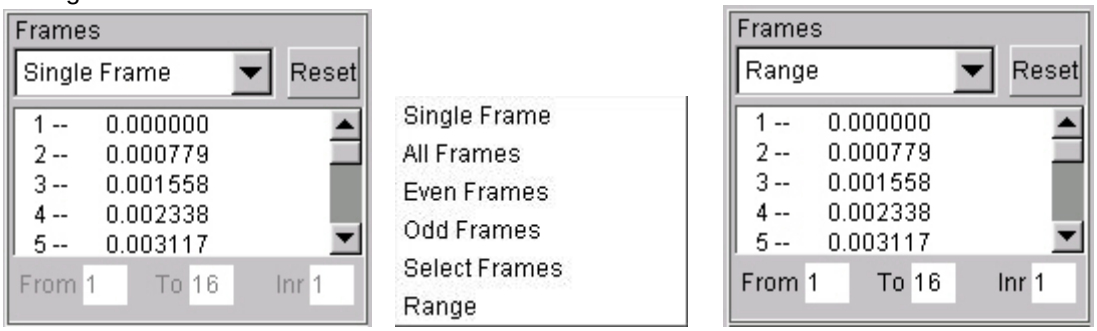


Figure 9.50 Frames Operations Figure 9.51 Frames Options Figure 9.52 Frame Range

- **FRAMES**
  The FRAMES window allows the user to select the individual frame(s) for the plot or animation. The frames selected are highlighted in blue. When frames are selected in other options, the corresponding frames are highlighted in the FRAME window.
- **SINGLE FRAME**
  This function allows user to pick single one frame, and eta/Post plots the deformation simultaneously.
- **ALL FRAMES**
  Once All Frames is picked, all frames in the list window are selected prepared for animation.
- **EVEN FRAMES**
  Once Even Frames is picked, even frames in the list window are selected and prepared for animation.
- **ODD FRAMES**
  Once Odd Frames is picked, odd frames in the list window are selected and prepared for animation.
- **SELECT FRAMES**
  Once Select Frames is picked, eta/Post allows user to select more than frames prepared for animation. The left mouse button picking works in conjunction with CTRL and SHIFT, whereupon the user can select any desired frames.
- **RANGE**
  The RANGE window allows the user multiple options for determining the frames to be animated shown as Figure 9.52. The RANGE option allows the user to input a select range of frames and the increment they will be run at. The input field for the RANGE option is not accessible until the option is selected.
  After inputs desired values, clicks ENTER to make the PLAY enabled.
- **RESET**
  This function clears all previously selected frames.

9.11.3 ANIMATION OPERATIONS

Description: This function allows the user to animate the selected frames shown as Figure 9.53

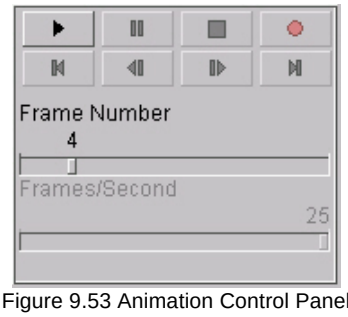


Figure 9.53 Animation Control Panel

- PLAY
  eta/Post stars to animate the selected frames. When the PLAY icon is selected, the

other three icons, PAUSE, STOP and EXPORT are active as seen in Figure 9.54.

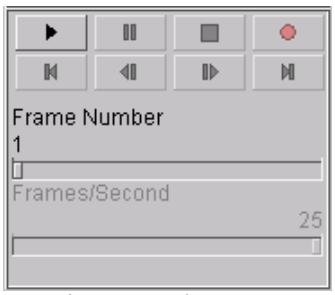


Figure 9.54 Play Status

- PAUSE
  Pause the animation. In this mode, the four other functions are enabled to use shown as Figure 9.55.

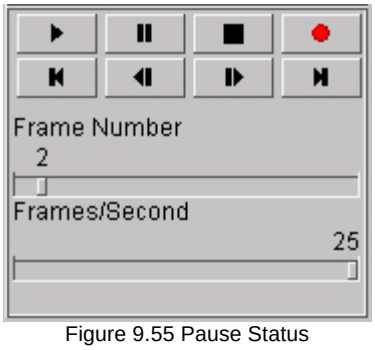


Figure 9.55 Pause Status

- FIRST FRAME display the first frame
- PREVIOUS FRAME display the previous frame
- NEXT FRAME display the next frame
- LAST FRAME display the end frame
- STOP Stops the animation.
- WRITE AVI AND E3D

  This function creates a .avi file or a .e3d file from current animation. An AVI file is a Microsoft multimedia file that provides a means to store a series of images for animation. An E3D file is an ETA 3D player file that provides a means to store the file for true 3D demo by eta/3D Player.
  Eta/Post outputs AVI or E3D files during the animation process only.
  To output an AVI file, first animate the result. During Animation, click the WRITE AVI icon. Eta/Post displays the WRITE FILE window for the user to choose a name and location for the AVI file.
  After entering the name and location of the file (SAVE button) in the WRITE FILE window, eta/Post displays the SELECT COMPRESSION FORMAT for the user to select the compression type and quality.
  Once the compression type and quality are determined, OK accepts and creates the file.
- **FRAME NUMBER**
  After pause the animation, this function allows the user to drag the slider to the desired frame and the frame will be plotted automatically.
  When play animation, the frame number is changed automatically according with the

animation.

- **FRAMES/SECOND**
  This function allows user to adjust the speed at which the animation is running by altering the number of frames per second. The number of frames is dragged to the desired position. To reduce the animation speed, the user must select a value lower than the current frame rate. Once the frame rate is selected, eta/Post will continue to use that rate until it is reset or the eta/Post session is ended.

CONTOUR

The functions in this menu animate the transient analysis results in real time, and are used to display the results of individual steps. The color bar, located on screen, displays the corresponding contour values. The location of the highest contour value on the model is labeled with an asterisk (*) and the lowest is labeled with a zero (0). The options are shown in Figure 9.56

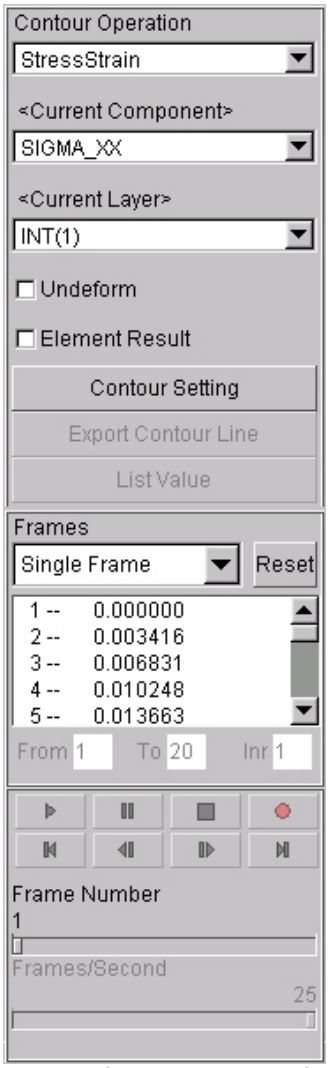


Figure 9.56 Animate Contour control window

9.11.4 SELECT COMPONENT

Description: There are STRESS/STRAIN and DISPLACEMENT. They determine the type of contour variable displayed during the plot/animation. The type of variable selected determines the type of plot component displayed in the CURRENT COMPONENT

CURRENT COMPONET

The CURRENT COMPONET WINDOW determines the type of component that will be displayed during the plot/animation. Only one component may be selected per plot/animation. The component types displayed in the window are determined by the types, STRESS/STRAIN or DISPALCEMENT.

If STRESS/STRAIN is selected, the supported components are listed below:
SIGMA_XX
SIGMA_YY
SIGMA_ZZ
SIGMA_XY
SIGMA_YZ
SIGMA_ZX
BEND_MONMENT_MXX
BEND_MONMENT_MYY
BEND_MONMENT_MZZ
SHEAR_RES_QXX
SHEAR_RES_QYY
NORMAL_RES_NXX
NORMAL_RES_NYY
NORMAL_RES_NXY
THICKNESS
EPSON_XX
EPSON_YY
EPSON_ZZ
EPSON_XY
EPSON_YZ
EPSON_ZX
ENERGY
MAX_VONMISES
MEAN_STRESS
PRIN_STRESS1
PRIN_STRESS2
PRIN_STRESS3
MAX_SHEAR_STRESS
PRIN_STRAIN1
PRIN_STRAIN2
THINNING
NORM_STRAIN

If DISPLACEMENT is selected, the supported components are listed below:
DISPLACEMENT_X
DISPLACEMENT_Y
DISPLACEMENT_Z
DISPLACEMENT_TOTAL

9.11.5 SELECT A LAYER

Description: Refer to NIP, number of integration point, in LS-DYNA menu.

9.11.6 UNDEFORM

Description: This function allows the resultant contour of the current frame projects on the undeformed, original blank surface. Please see the Figure 9.57 and 9.58 to compare the result.

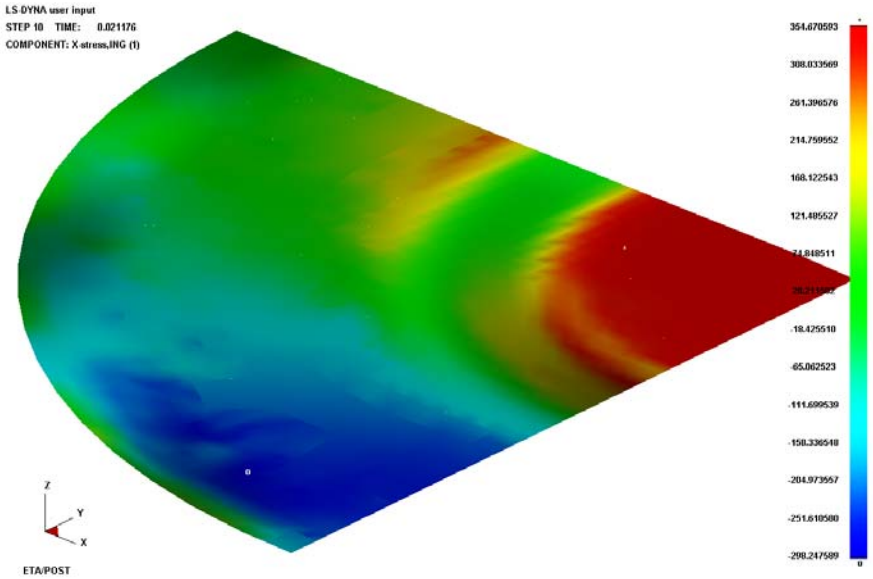


Figure 9.57 Project on Undeformed Blank

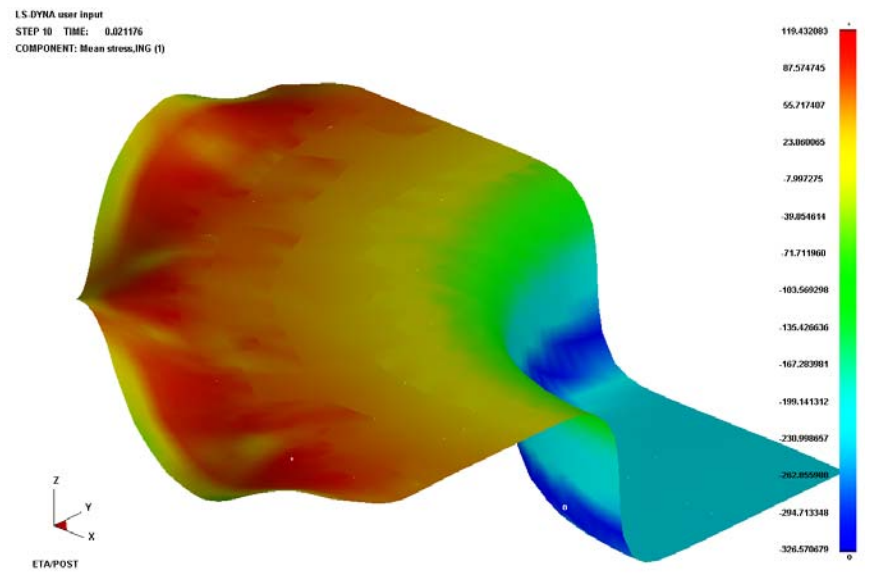


Figure 9.58 Current Frame Contour

9.11.7 ELEMENT RESULT

Description: This function determines whether the CONTOUR will be based on elements or nodes.

9.11.8 CONTOUR SETTING

Description: This function will start the Contour Bar Options Control Window shown as Figure 9.60. There are several options.

Usage:

- **REVERSE CONTOUR BAR**
  This function allows the user to reverse the color of contour bar for the contour plot. The result of the reverse contour bar is show in Figure 9.59.

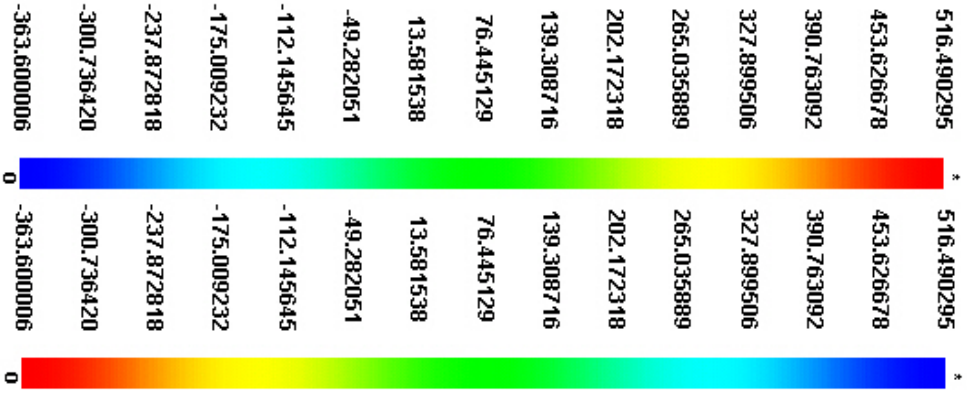


Figure 9.59 The Reverse contour Bar result.

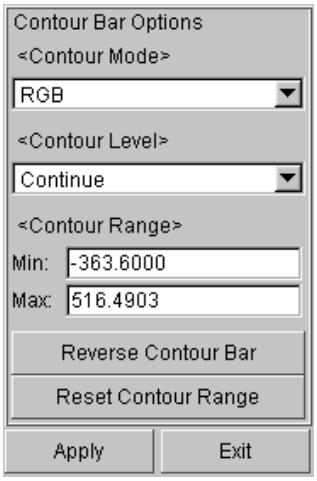


Figure 9.60 Contour Setting box.

- **RESET CONTOUR RANGE**
  This function resets the contour value range to the default maximum/minimum values. If the value is more than the maximum, the contour is displayed in dark colour of the top bar, and if the value is less than the minimum value, the contour is displayed in dark color of the bottom bar. Please see the Figure 9.61

Note: Contour Range is only accessible after plotting. Once the user has made a plot and selected SET

CONTOUR RANGE, the lower range and upper range are entered via the DAPT INPUT field.

The APPLY button is then selected and the model will be re-plotted with new range.

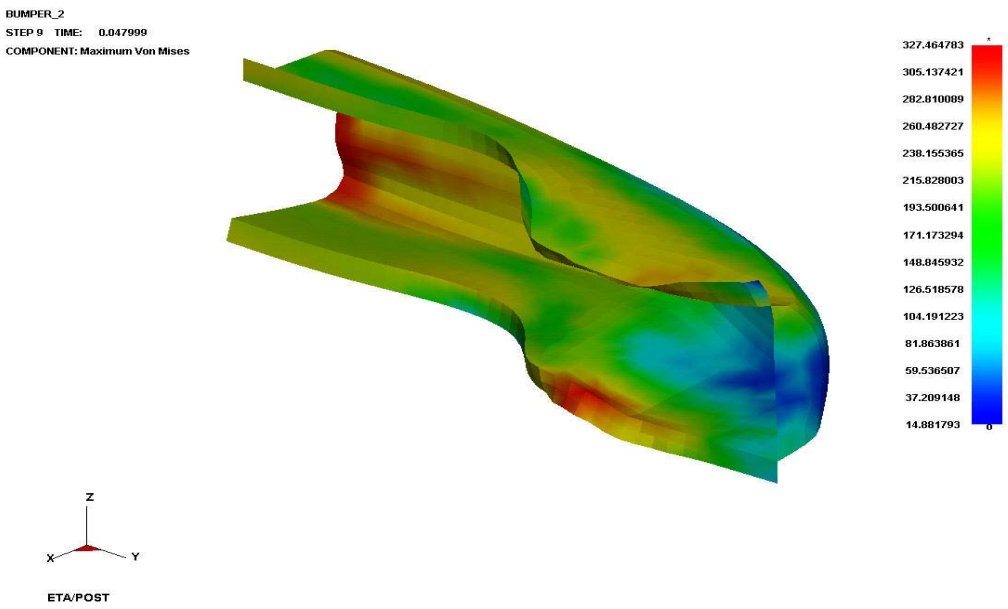


Figure 9.61 An Example for Contour Range

- **CONTOUR MODE**
  There are four types of colour.

  RGB (red, green, blue)
  RGBM (red, green, blue, magenta)
  RG (red, green)
  GRAY

- **CONTOUR LEVEL**
  The contour level can be set from 2-20 levels. The program defaults to CONTINUE. If the user sets the contour level with any value 2-20, the contour is displayed discretely as contour line. Please see the Figure 9.62.

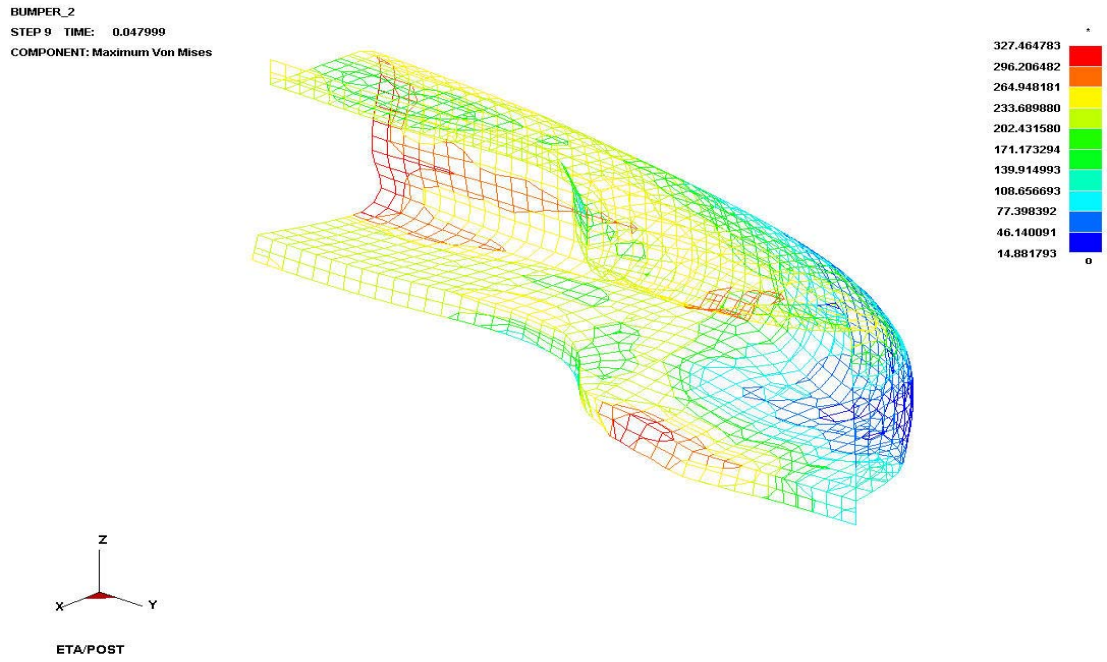


Figure 9.62 An Example for Contour Line

9.11.9 EXPORT CONTOUR LINE

Description: This function allows user to export the contour line (when the CONTOUR LEVEL is not CONTINUE) into VPG Line Data format file.

Usage: eta/Post prompts the user to enter the file name. After determining the file name and clicking SAVE button, the contour lines will be saved in the given file.

9.11.10 LIST VALUE

Description: This function lists the contour value of selected nodes or selected element (if ELEMENT RESULT is toggled on). For example, list node value as below step:

Usage:

1. Activate the LIST VALUE function in the CONTOUR CONTROL WINDOW.

2. The Control Option window is displayed with Select Node Option shown as Figure 9.63.

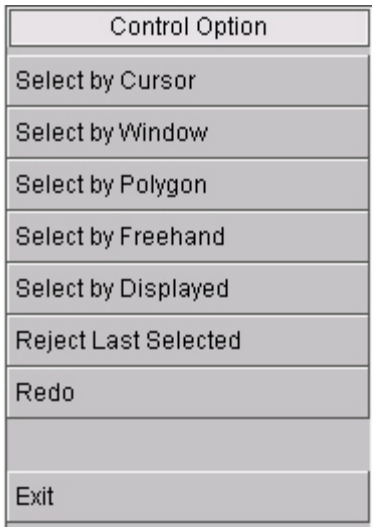


Figure 9.63 Select Node Control Option

3. Select the desired node to be listed. The default setting is by mouse pick. The user can also select nodes by dragging window, polygon or free hand region.

4. Once the node is selected, the highest three nodes are listed at the down left corner of the GRAPHICS DISPLAY WINDOW shown as Figure 9.64.

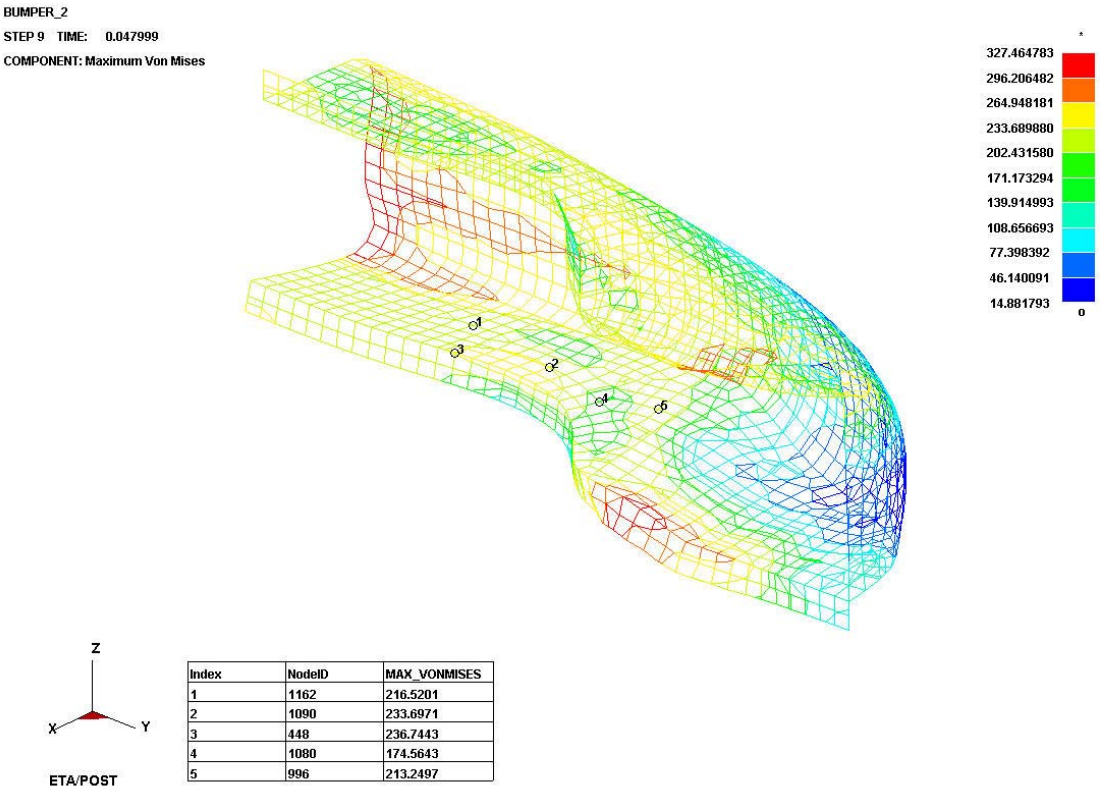


Figure 9.64 List Node Value

5. At the same time, another Control Option Window appears shown as Figure 9.65 The functions control the number and position of nodes listed, and which nodes

should be listed.

Figure 9.65 List Value Control Option

HIGHEST
The set of nodes with highest value are listed.

NEXT HIGHEST
The set of nodes with next highest value are listed.

NEXT LOWEST
The set of nodes with next lowest value are listed.

LOWEST
The set of nodes with lowest value are listed.

SET LIST NUMBER
The function prompts a data control window that allows the user to enter the number of the listed nodes.

SET LIST POSTION
This function allows the user to place the list window by the cursor pick.

Other functions are common with those the SECTION DEFORMATION.

VECTOR

The functions in this menu animate the transient analysis results in real time, and are used to display the results of individual steps with the vector. The options are shown in Figure 9.66.

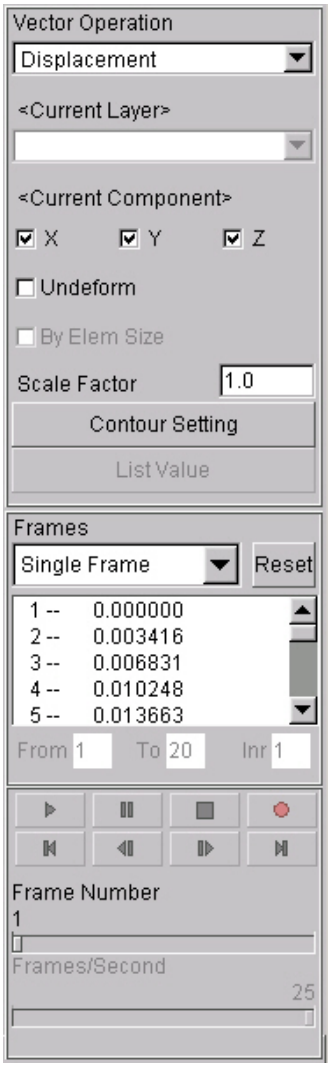


Figure 9.66 Vector Operation control window

9.11.11 SELECT COMPONENT

Description: There are four types of component, DISPLACEMENT, VELOCITY, ACCELERATION, STRESS and STRAIN in the function.

Usage: When selecting DISPLACEMENT, VELOCITY or ACCELERATION, the CURRENT LAYER is disabled, and the CURRENT COMPONENT is enabled used. The user can select them in any combination.

If STRESS or STRAIN is selected, the CURRENT LAYER is enabled and the CURRENT COMPONENT is disabled. There are MIDDLE, TOP and BOTTOM available in the CURRENT LAYER.

9.11.12 BY ELEMENT SIZE AND SCALE FACTOR

Description: This function is used to alter the size of the vector. **BY ELEMENT SIZE** means

the vector size is based on the element size.

Usage: **SCALE FACTOR** allows the user to enter a percentage of the maximum magnitude contained the model.

Note: When BY ELEMENT SIZE is toggled on, the SCAL FACTOR is disabled.

Note: Reset Contour Range is in the VECTOR. The value more than the maximum, the vector is displayed in red and the value less than the minimum, while the vector is displayed in blue.

9.12 GRAPH

The GRAPH function is a more independent comparatively module than other functions of eta/Post-Processor listed above. This function enables the user to visualize time history result from LS-DYNA analysis with graph format. Moreover, the GRAPH function offers a wide range of tools to help the user better understand and convey the results. Features include the ability to manipulate the display area's settings (labels, colors, etc) and a host of advanced filtering techniques (FIR, SAE, Butterworth, averaging, etc.). The detailed description of available function in GRAPH is given in the following sections.

The GRAPH function can be active by clicking the GRAPH icon on toolbar in despite of if the user have loaded a model file to current database or not. Then the result file load control window will pop up on the right of screen for the user to load the time history result file. The control window is shown in Figure.9.67.

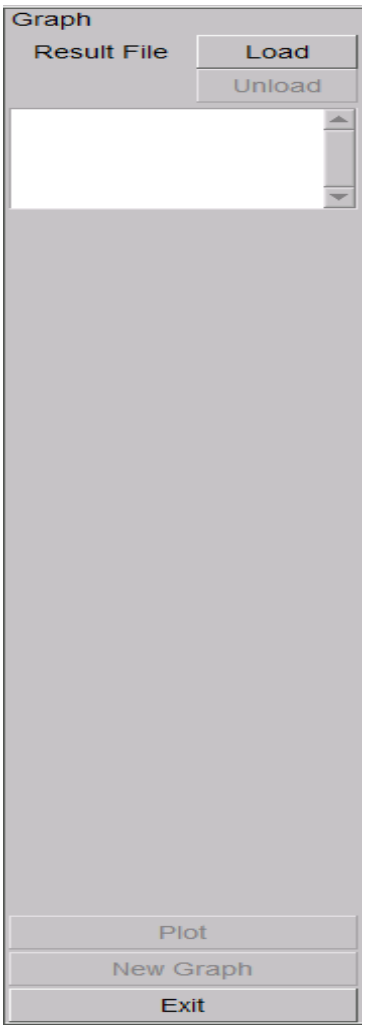


Figure 9.67 The Initial Graph Window

9.12.1 LOAD RESULT FILE

Description: The function allows the user to read result files form LS-DYNA analysis into the current database. The user has the ability to input LS-DYNA ASCII, State and Time data formats. Click the **Load** button and the program pops up the Select File dialog box for user to select a database file. The figure is shown in Figure 9.68.

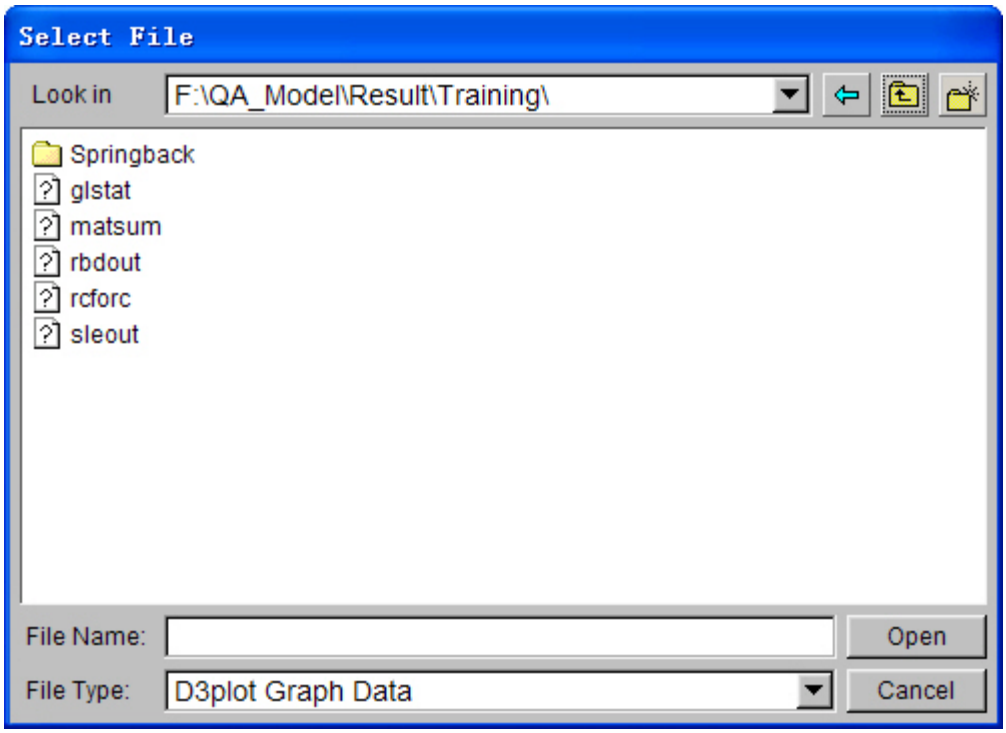


Figure 9.68 Select File dialog box.

- D3PLOT GRAPH DATA

Description: There are some kinds of D3plot graph data format supported by eta/Post-Processor. Those file formats including:

ABSTAT	Airbag statistics
BNDOUT	Boundary nodal forces
DEFGEO	Deformed geometry
DEFORC	Discrete elements
ELOUT	Element data
GCEOUT	Contact element resultants
GLSTAT	Global data
JNTFORC	Joint force file
MATSUM	Material energies
NCFORC	Contact interface forces
NODFOR	Nodal force
NODOUT	Nodal point data
RBDOUT	Rigid body data
RCFORC	Resultant interface forces
RWFORC	Wall forces
SBTOUT	Seatbelt output
SECFORC	Cross section forces

SLEOUT	Sliding interface energy
SPCFORC	Single point constraint (SPC) reaction forces
SSSTAT	Subsystem statistics
SWFORC	Spotweld rivet forces
TPRINT	Temperature output

- AIRBAG STATISTICS (ABSTAT)

Description: The user can plot airbag statistic data from the **"abstat"** file. The following types are available:

VOLUME
PRESSURE
INTERNAL ENERGY
DM/DT IN
GENSITY
DM/DT OUT
TOTAL MASS
GAS TEMPERATURE

-BOUNDARY NODAL FORCES (BNDOUT)

Description: The user can plot boundary nodal force time history data from the "**bndout"** file. The following types are available:

XFORCE
YFORCE
ZFORCE
ENERGY

-DEFORMED GEOMETRY (DEFGEO)

Description: The user can plot the deformed geometry data from the **"defgeo**" file.

-DISCRETE ELEMENTS (DEFORC)

Description: This user can plot the discrete element time history data from the "**deforc**" file. The following types are available:

X-FORCE (MOMENT)
X-FORCE (MOMENT)
X-FORCE (MOMENT)

RESULTANT FORCE (MOMENT)

-ELEMENT DATA (ELOUT)

Description: The user can plot element force data from the **"elout"** file.

-CONTACT ELEMENT RESULTANTS (GCEOUT)

Description: The user can plot contact element resultant data from the "**gceout"** file. The following types are available:

XFORCE
YFORCE
ZFORCE
FORCE MAGITUDE
X MOMENT
Y MOMENT
Z MOMENT
MOMENT MAGNITUDE

-GLOBAL DATA (GLSTAT)

Description: The user can plot the global time history data from the **"glstat"** file. The following types are available:

TIME STEP
TOTAL ENERGY
TOTAL/INITIAL ENERGY
ENERGY RATIO
KINETIC ENERGY/ERODED KINETIC ENERGY
INTERNAL ENERGY/ERODED INTERNAL ENERGY
SPRING & DAMPER EMERGY
HOURGLASS ENERGY
SYSTEM DAMPING ENERGY
SLIDING INTERFACE ENERGY
EXTERNAL WORK
TIME PER ZONE CYCLE
NUMBER OF SHELL ELEMENT
STEP SIZE
GLOBAL X VELOCITY
GLOBAL Y VELOCITY
GLOBAL Z VELOCITY

-JOINT FORCE FILE (JNTFORC)

Description: The user can plot the joint force data from the **"jntforc"** file. The following types are available:

X_FORCE
Y_FORCE
Z_FORCE
X_MOMENT
Y_MOMENT
Z_MOMENT
RESULTANT_FORCE
RESULATANT_MOMENT

-MATERIAL ENERGIES (MATSUM)

Description: The user can plot the material energy time history data from the **"matsum"** file. The following types are available:

INTERNAL ENERGY
KINETIC ENERGY
X MOMENT
Y MOMENT
Z MOMENT
X RIGID BODY VELOCITY
Y RIGID BODY VELOCITY
Z RIGID BODY VELOCITY

-CONTACT INTERFACE FORCES (NCFORC)

Description: The user can plot the contact interface force data from the **"ncforc"** file. The following types are available:

X FORCE
Y FOECE
Z FORCE
PRESSURE
X COORDINATE
Y COORDINATE
Z COORDINATE

-NODAL FORCE (NODFOR)

Description: The user can plot the nodal force data from the **"nodfor"** file.

-NODAL POINT DATA (NODOUT)

Description: The user can plot the nodal point data from the **"nodout"** file. The following types are available:

X DISPLACEMENT
Y DISPLACEMENT
Z DISPLACEMENT
X VELOCITY
Y VELOCITY
Z VELOCITY
X ACCELERATION
Y ACCELERATION
Z ACCELERATION

-RIGID BODY DATA (RBDOUT)

Description: The user can plot the rigid body time history data from the "**rbdout"** file. The following types are available:

GLOBAL X COORDINATE
GLOBAL Y COORDINATE
GLOBAL Z COORDINATE
GLOBAL X DISPLACEMENT
GLOBAL Y DISPLACEMENT
GLOBAL Z DISPLACEMENT
GLOBAL X ROT DISPLACEMENT
GLOBAL Y ROT DISPLACEMENT
GLOBAL Z ROT DISPLACEMENT
GLOBAL X VELOCITY
GLOBAL Y VELOCITY
GLOBAL Z VELOCITY
GLOBAL X ROT VELOCITY
GLOBAL Y ROT VELOCITY
GLOBAL Z ROT VELOCITY
GLOBAL X ACCELERATION
GLOBAL Y ACCELERATION
GLOBAL Z ACCELERATION
GLOBAL X ROT ACCELERATION
GLOBAL Y ROT ACCELERATION
GLOBAL Z ROT ACCELERATION
DVECTOR ROW_1-A
DVECTOR ROW_1-B
DVECTOR ROW_1-C
DVECTOR ROW_2-A
DVECTOR ROW_2-B
DVECTOR ROW_2-C
DVECTOR ROW_3-A
DVECTOR ROW_3-B
DVECTOR ROW_3-C
LOCAL A DISPLACEMENT
LOCAL B DISPLACEMENT
LOCAL C DISPLACEMENT
LOCAL A ROT DISPLACEMENT
LOCAL B ROT DISPLACEMENT
LOCAL C ROT DISPLACEMENT
LOCAL A VELOCITY
LOCAL B VELOCITY
LOCAL C VELOCITY
LOCAL A ROT VELOCITY
LOCAL B ROT VELOCITY
LOCAL C ROT VELOCITY
LOCAL A ACCELERATION
LOCAL B ACCELERATION
LOCAL C ACCELERATION
LOCAL A ROT ACCELERATION
LOCAL B ROT ACCELERATION
LOCAL C ROT ACCELERATION

-RESULT INTERFACE FORCES (RCFORC)

Description: The user can plot the resultant interface data from the **"rcforc"** file. The following types are available:

X FORCE
Y FORCE
Z FORCE
MASS

-WALL FORCES (RWFORC)

Description: The user can plot the rigid wall time history data from the **"rwforc"** file. The following types are available:

NORMAL FORCE
X FORCE
Y FORCE
Z FORCE

-SEATBELT OUTPUT (SBTOUT)

Description: The user can plot the seatbelt output data from the **"sbtout"** file. The following types are available:

SEATBELT
SLIPRING
RETRACTOR

-CROSS SECTION FORCE (SECFORC)

Description: The user can plot the cross section time history data from the **"secforc"** file. The following types are available:

X FORCE	X MOMENT
Y FORCE	Y MOMENT
Z FORCE	Z MOMENT
X CENTROID	TOTAL FORCE
Y CENTROID	TOTAL MOMENT
Z CENTROID	AREA

-SLIDING INTERFACE ENERGY (SLEOUT)

Description: The user can plot the interface energy data from the **"sleout"** file. The following types are available:

TOTAL SLAVE SIDE

TOTAL MASTER SILE
TOTAL ENERGY

-SPC REACTION FORCES (SPCFORC)

Description: The user can plot the SPC reaction force data from the **"spcforc"** file. The following types are available:

X FORCE
Y FORCE
Z FORCE
X MOMENT
Y MOMENT
Z MOMENT

-SUBSYSTEM STATISTICS (SSSTAT)

Description: The user can plot the subsystem statistic data from the "**ssstat"** file. The following types are available:

KINETIC ENERGY GLOGAL
KINETIC ENERGY SUBSYSEM 1
KINETIC ENERGY SUBSYSEM 2
KINETIC ENERGY SUBSYSEM 3
KINETIC ENERGY SUBSYSEM 4
INTERNAL ENERGY GLOBAL
INTERNAL ENERGY SUBSYSTEM 1
INTERNAL ENERGY SUBSYSTEM 2
INTERNAL ENERGY SUBSYSTEM 3
INTERNAL ENERGY SUBSYSTEM 4
KINETIC ENERGY RATIOS 1
KINETIC ENERGY RATIOS 2
KINETIC ENERGY RATIOS 3
KINETIC ENERGY RATIOS 4
INTERNAL ENERGY RATIOS 1
INTERNAL ENERGY RATIOS 2
INTERNAL ENERGY RATIOS 3
INTERNAL ENERGY RATIOS 4
X MOMENT 1
X MOMENT 2
X MOMENT 3
X MOMENT 4
Y MOMENT 1
Y MOMENT 2
Y MOMENT 3
Y MOMENT 4
Z MOMENT 1
Z MOMENT 2
Z MOMENT 3
Z MOMENT 4

-SPOTWELD RIVET FORCES (SWFORC)

Description: The user can plot the spotweld rivet force data from the **"swforc"** file. The following types are available:

AXIAL
SHEAR

-TEMPERATURE OUTPUT (TPRINT)

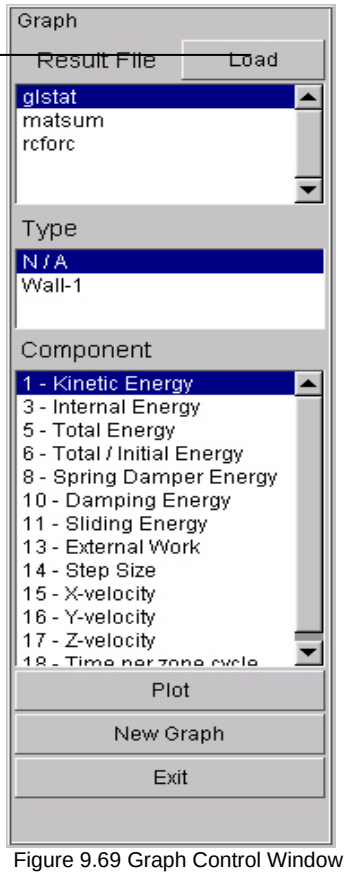


Figure 9.69 Graph Control Window

Description: The user can plot the temperature output data from the **"tprint"** file. The following types are available:

HEAT GENERATION
TOTAL HEAT GENERATION
CHANGE INTERNAL ENERGY
INTERNAL ENERGY

NOTE: The available data type depends on the data file.

Usage: Select a file for the **Select File** dialog box and click **Open** to load the desired database to the Post-Processor. There are many different kinds of result files from LS-DYNA analysis that can be read in and the numbers of files which are depend on the type of analysis and the setting in preprocessor are not the same every time. Moreover, the user can load several files in the current database though repeating the operation above. The Graph control windows will pop up as Figure 9.69.

The number of List Boxes is not the same every time and depends on the result file type. The detailed description about those functions is given in the following section.

9.12.2 RESULT FILE LIST BOX

Description: The Result File list box displays the name of result files that have been loaded in the current database. The user can select the desired name listing in the result file list box and from which to retrieve the intend data.

9.12.3 TYPE LIST BOX

Description: The items list the Type list box depends upon the result file selected on the Result file list box and only the actual contained data types will be displayed and the list box item may change depending upon the project. For example in Figure 9.67, there are two types N/A and Wall-1.

9.12.4 COMPONENT LIST BOX

Description: The items list on Component box are not only depends upon the selected file type on Result file list box also depends upon the selected type on type list box. Moreover, the same as the item on type list only the actual contained data types will be displayed and the list box item may change depending upon the items specified in the analysis.

9.12.5 PLOT

Description: This button allows the user to plot the selected item on Component list box with graph in current Curve window. The example is shown on Figure 9.70.

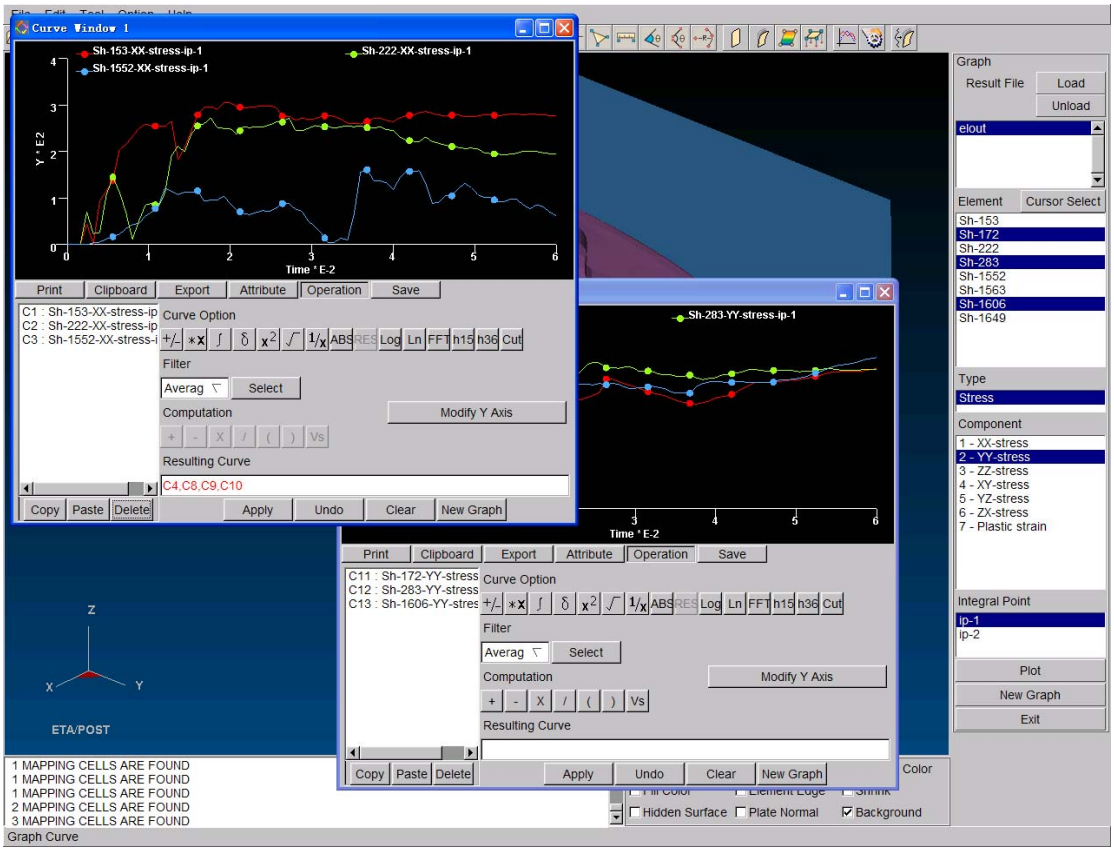


Figure 9.70 the eta/Post-Processor Interface Window.

9.12.6 CURVE OPERATION TOOLBAR

Description: The Curve Operation Toolbar that is shown in Figure 9.71.

Figure 9.71 The Curve operation toolbar.

In the Curve operation toolbar there are six buttons can be used and the detailed functional description is given in the following section.

Usage:

- **PRINT**

This function enables the user to print or save the content of the current curve window.

- **CLIPBOARD**

This function enables the user to copy the content of the current curve window to the clipboard and can be used for other text edit tool such as MS-Word. Copy to Clipboard is the standard Windows method of transferring data between a source and a destination. Copy to Clipboard is a system service shared by the entire Windows session. Using this function the user can copy a graph image directly to other Microsoft programs as a bitmapped image provided that the destination program supports a bitmapped image paste. A demonstration as to inserting an image form clipboard to Microsoft Word is illustrated in figure 9.72.

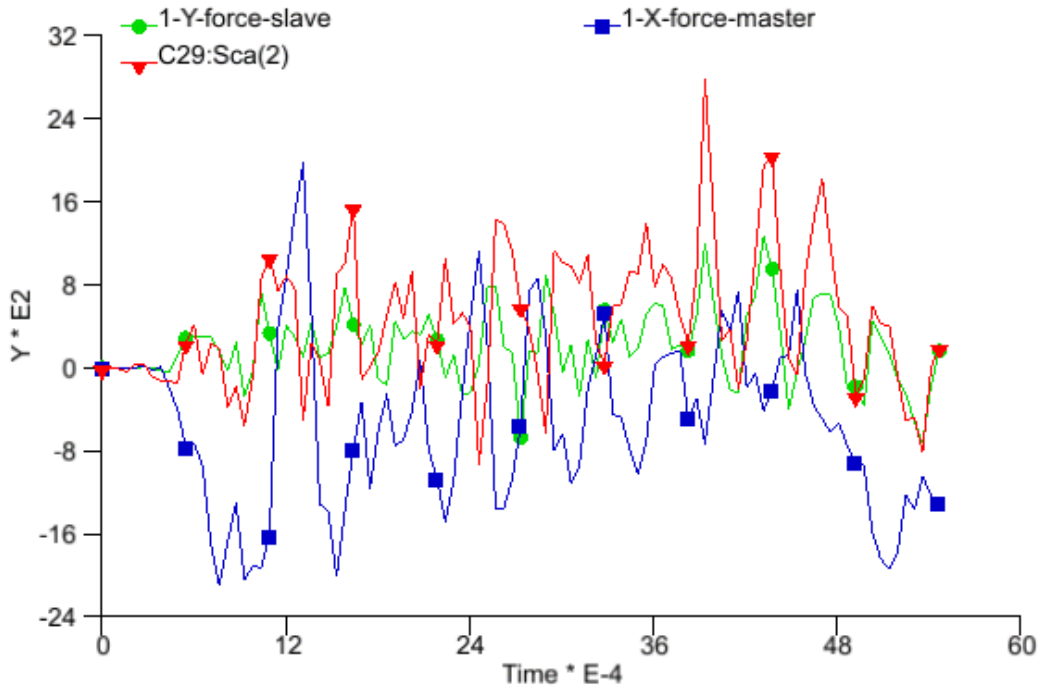


Figure 9.72 The demonstration on inserting an image form clipboard to MS-Word

- **EXPORT**

This function enables the user to export and save the current window curves in a file with the .CUR extension. The Select File dialog box will display and prompt the user to select the desired name.

- **ATTRIBUTE**

This function enables the user to change the attribute of the curve. Clicking the Attribute button the pop up window will be displayed under the curve window. The pop up window is shown in figure 9.73.

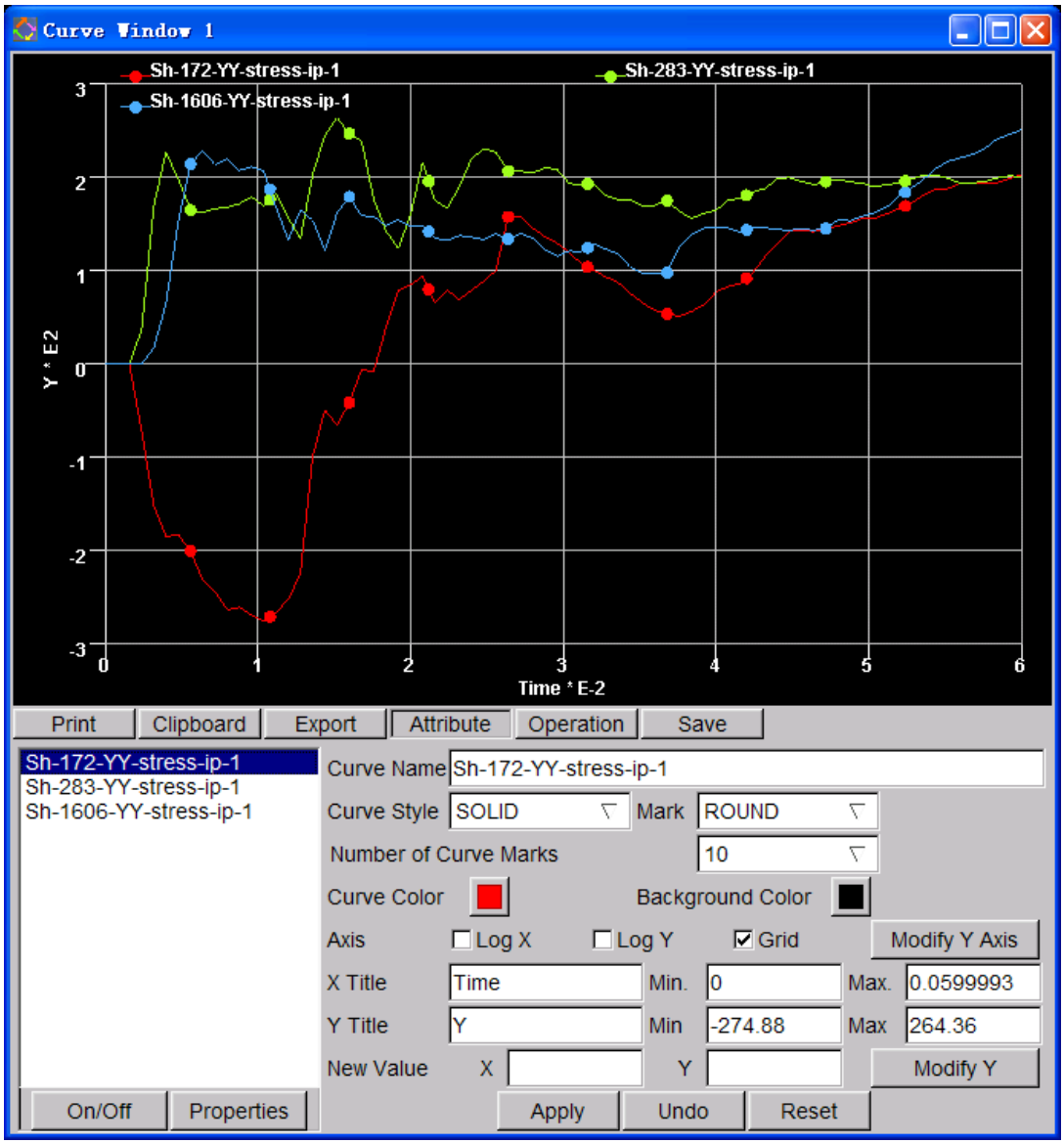


Figure 9.73 The Graphing Attribute menu.

There are many options for the user to change the attribute of the selected curve. On the left of the patulous window the name of current curve is listed. The user should select the intended name of curve to change the attribute.

9.12.6.1 CURVE NAME

Description: This function offers an option for the user to change the name of the curve.

9.12.6.2 CURVE STYLE

Description: This function offers an option for the user to change the displaying type of curve. There are four kinds of display type available.

SOLID

DASH
DOT
DASH DOT

Usage: The default type is SOLID and the user can press the button on the right of the text box to select a desired type. Then click the Apply button to change the setting.

9.12.6.3 CURVE MARK

Description: This function offers an option for the user to change the type of curve mark. There are six kinds of curve mark type available.

ROUND
SQUARE
UP TRIANGULAR
DOWN TRIANGULAR
DIAMOND
NO MARK

Usage: The user can press the button on the right of the text box to select a desired type. Then click the Apply button to change the setting.

9.12.6.4 CURVE COLOUR

Description: This function offers an option for the user to change the colour of curve.

Usage: The colour of the current curve is displayed on the button. The user can click this button and the colour panel will pop up and prompt the user to select the desired colour for the curve. The colour panel is shown in figure 9.74. If don't want to change the colour after the colour panel have popped up, the user can click anywhere else on the screen to cancel the colour select. Then click the Apply button to change the setting.

Figure 9.74The Curve Color Palette

9.12.6.5 BACKGROUND COLOR

Description: This function offers an option for the user to change the background colour of the curve window. The default background colour of the curve window is black and the colour is displaying on the button.

Usage: The user can click this button and the colour panel will pop up and prompt the user to select the desired colour for the curve. The colour panel is shown in figure 9.72. If don't want to change the colour after the colour panel have popped up, the user can click anywhere else on the screen to cancel the colour select. Then click the Apply button to change the setting.

9.12.6.7 AXIS OPERATION

Description: The functions of the Axis Operation enable the user to modify the attributes of the axis of coordinate, including the Log X and Log Y Grid display and the change in height of Y axis.

9.12.6.8 LOG X and LOG Y (toggle)

Description: This function enables the user to change the X and Y coordinate values to logarithmic values.

9.12.6.9 GRID

Description: This function enables the user to display grid on the curve window.

9.12.6.10 AXIS HEIGHT

Description: This function enables the user to specify the height of y-axis on the curve window through clicking with left mouse on the area of positive orientation of y-axis. The user can re-click the Modify button to cancel this operation.

9.12.6.11 X TITLE

Description: This function offers an option for the user to change the title of X axis (horizontal). The x title will be displayed under the X axis.

9.12.6.12 Y TITLE

Description: This function offers an option for the user to change the title of Y-axis (vertical). The Y title will be displayed under the Y axis.

9.12.6.13 MIN. MAX

Description: This function offers an option for the user to change the range of axis of coordination.

9.12.6.14 NEW VALUE

Description: This function allows the user to modify the y value of selected point. Clicking a point on a curve with left mouse a cross is displaying on the selected point and the x and y value of coordinate of the point is displaying on the corresponding text boxes. The X text box is inactive and can't be modified. The Y text box is active and the user can modify the value on it.

9.12.6.15 OPERATION

Description: This OPERATION allows the user to modify selected curve. There are some kinds of operations on curve objects. These operations belong to two categories: curve data operation such as integration, different, square, root and so on and curve object operations such as Copy, Delete and Paste. One or more curves must be selected before modify the data of the curve. Clicking the Operation button the pop up window will display under the curve window. The pop up window is shown in figure 9.75.

Usage:

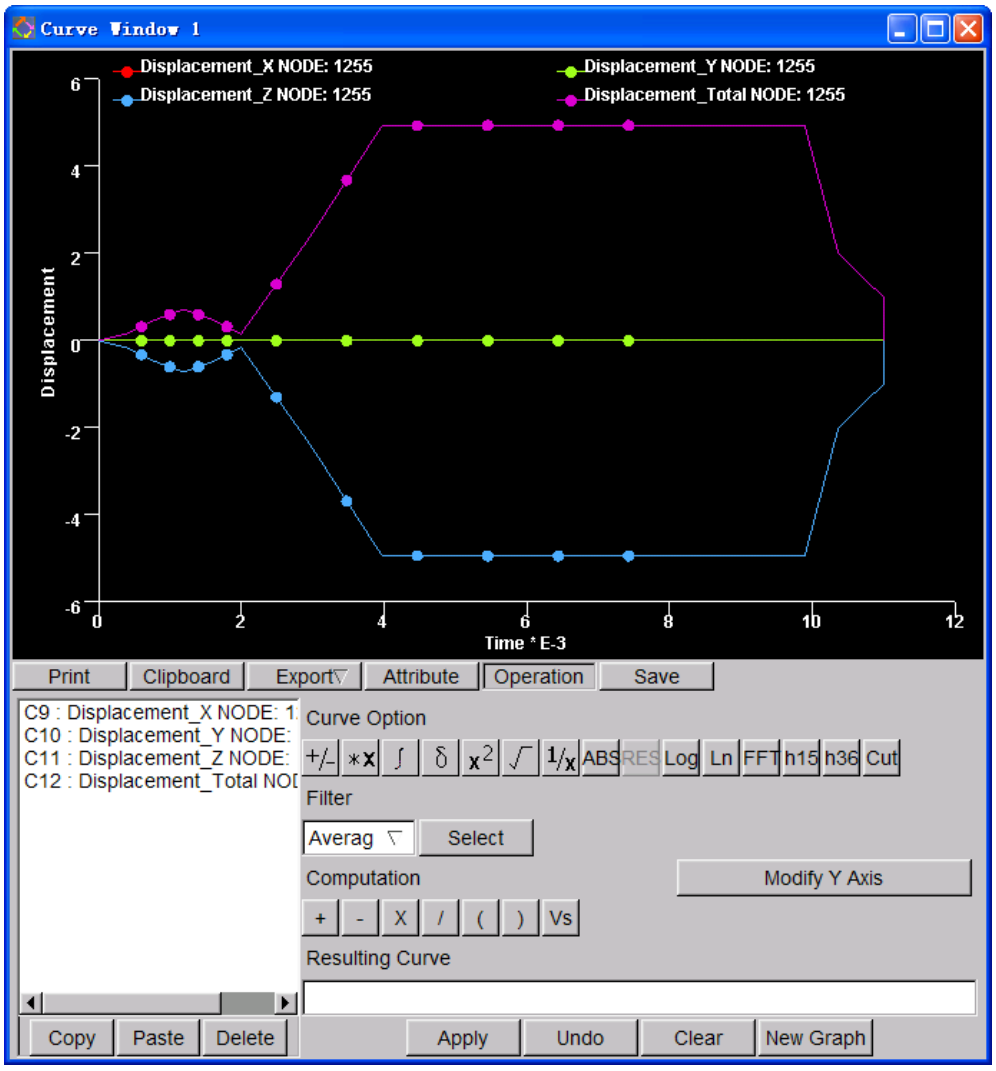


Figure 9.75 The Curve Operation Window

There is a curve list box on the left of the Operation window. The curve names are list on this list box. The user can select one or several curve name(s) through press the left mouse and move it.

CURVE OBJECT OPERATION

- **COPY**

This function allows the user to copy the selected curve(s).

- **PASTE**

This function allows the user to paste the copied curve(s) to the current curve window.

NOTE: The user cannot paste the curve to other software such as MS-Word since the format that copy from curve is the inner format of eta/Post-Processor. If the user wants to paste to other software, using the Clipboard function is available way.

- **DELETE**

This function allows the user to delete the selected curve(s) from the current curve window.

9.12.6.16 CURVE DATA OPERATION

CURVE OPTION

The Curve Options includes 9 kinds of operation for the curve data.

NEGATIVE

This function allows the user to change the Y-axis value of selected curves through multiplying -1.

SCALE

This function allows the user superimposes the scale curve that scales the Y-axis value of selected curves through multiplying a given Y scale factor.

INTEGRATIE

This function allows the user to superimpose the integral of the curve on the graph which the Y-axis value is equal to the area under the selected curve from start point to the current X point.

DIFFERENTIAL

This function allows the user to superimpose the derivation of the curve on the graph which the Y-axis value is equal to the slope of the curve's tangent on corresponding X point.

SQUARE

This function allows the user to superimpose the square of the curve on the graph where the Y-axis value is equal to the original Y-axis value multiplying by itself.

ROOT

The function allows the user to superimpose the root of the curve on the graph, where the Y-axis value is equal to the root of the absolute value of the original Y-axis value.

RECIPROCAL

This function allows the user to superimpose the reciprocal value of the curve on the graph, where the Y-axis value is equal to the reciprocal value of the original Y-axis value.

LOGARITHM

This function allows the user to superimpose the logarithm of the curve on the graph, where the Y-axis value is equal to the logarithm value of the original Y-axis value.

NAPIERIAN LOGARITHM

This function allows the user to superimpose the Napierian logarithm of the curve on the graph, where the Y-axis value is equal to the Napierian logarithm value of the original Y-axis value.

ABSOLUTE VALUE

This function allows the user to plot the absolute value of the Y values of the selected curve.

RESULTANT

This function allows the user to find the square root of the sum of the square of the Y values of the selected curves. The new value is $Y_{new} = \sqrt{Y_1^2 + Y_2^2 + Y_3^2}$. The function is used to plot the vector length from three curves containing the X, Y and Z component. For a two-dimensional vector, only two curves need to be selected. This function will be activated after the user selects more than one curve.

FFT

This function allows the user to convert curve data from time domain to frequency domain response for signal processing analysis. The FFT is a fast algorithm for computing the Discrete Fourier Transform (DFT). The DFT is a basic operation to transform an ordered sequence of data samples from a signal, usually in a time-domain into the frequency-domain. The spectral information about the signal is then represented explicitly. There are various implementation of FFT when the samples are not of power of two. The algorithms adopted in eta/Post can deal with both cases, whether the number of samples is a power of two or not.

HIC 15

This function enables the user to calculate the maximum Head Injure Criteria when the impact time is less than 15 milliseconds. This is a special function used for eta/VPG

HIC 36

This function enables the user to calculate the maximum Head Injure Criteria when the impact time is less than 36 milliseconds. This is a special function used for eta/VPG

CUT

This function will cut the curve from a time to another time

9.12.6.17 FILTERS

Description: In this section an introduction on digital filter is presented, including the filter design, usage and result interpretation. There are four different filters implemented in OPERATION. They are:

- Average Filter
- Butterworth Filter
- Finite Impulse Response (FIR)
- SAE Filter

9.12.6.18 AVERAGE

Description: Average allows the user to smooth the curve through averaging the value of a point with a number of surrounding points defined by user. This averaging of the points can make the curve become smoother.

Usage: Select the Average type from the pop up list window via clicking the down triangular button below the FILTER.

Clicking the Select button the DATA CONTROL windows will pop up and prompt the user to input the number of the points for averaging as Figure 9.76 is shown. The user may enter the appropriate coefficient in the input box, which displays the default value of 10.

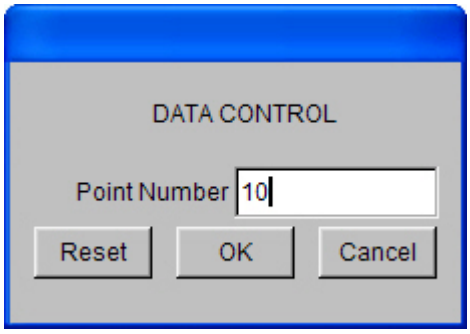


Figure 9.76 DATA CONTROL window of AVERAGE filter

After the user enters the appropriate coefficient, presses OK to accept the input value and exit the DATA CONTROL window.

Press Apply to display the filter result.

9.12.6.19 FIR FILTER

Description: This function allows the user to smooth the curve according to the finite impulse response (FIR) filter specifications.

Usage: Select the FIR type from the pop up list window via clicking the down triangular button below the FILTER.

Clicking the Select button the DATA CONTROL windows will pop up and prompt the user to input the corresponding parameters as Figure 9.77 shown. The user may enter the appropriate coefficient parameters in the DATA CONTROL windows which displays the default values for:

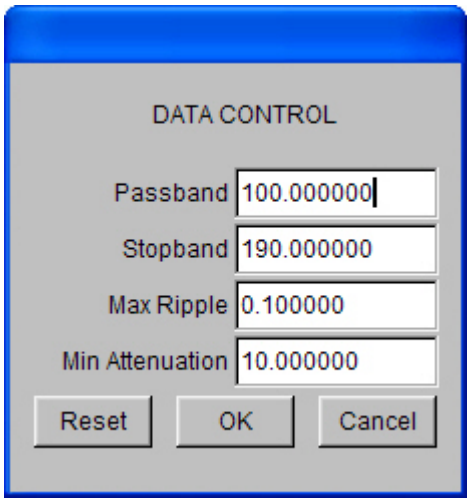


Figure 9.77 DATA CONTROL of FIR filter

9.12.6.20 PASSBAND FREQUENCY

Description: The maximum passband frequency is determined through the equation: $Passband_{user}$ = (Total Number of Points on Graph/Time Duration). The default value is 10% of the passband frequency.

9.12.6.21 STOPBAND FREQUENCY

Description: Stopband frequency is formulated; Stopband = $Passband_{user}$ + ($Passban_{max}$ - $Passband_{user}$). The default value depends on the $Passband_{user}$.

9.12.6.22 MAXIMUM PASSBAND RIPPLE (MAX PASSBRIPP)

Description: Passband ripple value is between 0.0 and 1.0 dB.

9.12.6.23 MINIMUM STOPBAND ATTENUATION (MIN.ATT.1.0+ DB)

Description: Stopband attenuation must be greater than 1.0 dB. The stopband frequency and maximum passband ripple should not be too close to either the given low- or high-end limits. Attenuation factors commonly range between 10 and 50. It is recommended that the user choose a passhand frequency that is approximately 5 to 20% of the number. Too small a number will cause a computational error and too large a number will cause the program to stall.

Usage: After the user enters the appropriate parameters, clicks the OK and accept the input parameters and exit the DATA CONTROL window.

Select Apply to display the filter result.

9.12.6.24 SAE FILTER

Description: This function allows the user to smooth curves according to the Society of Automotive (SAE) filter specification.

Usage: Select the SAE type from the pop-up list window via clicking the down triangular button below the FILTER.

Clicking the **Select** button the DATA CONTROL windows will pop up and prompt the user to input the Cutoff value as Figure 9.78 shown. The user may enter the appropriate value in the DATA CONTROL windows which displays the default values for:

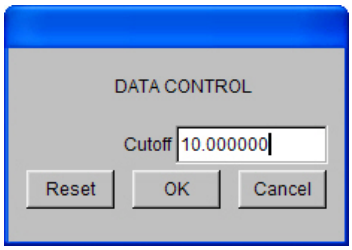


Figure 9.78 DATA CONTROL window for SAE Filter

After the user enters the appropriate value, clicks the OK and accept the input parameters and exit the DATA CONTROL window.

Select Apply to display the filter result.

9.12.6.25 BUTTERWORTH FILTER

Description: This function allows the user to smooth the curves according to the Butterworth filter specifications.

Usage: Select the Butterworth type from the pop-up list window via clicking the down triangular button below the FILTER.

Clicking the Select button the DATA CONTROL windows will pop up and prompt the user to input the corresponding parameters as Figure 9.79 shown. The user may enter the appropriate coefficient parameters in the DATA CONTROL windows which displays the default values for:

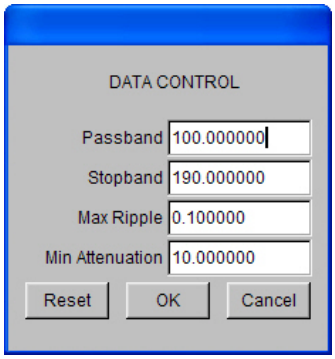


Figure 9.79 DATA CONTROL window for Butterworth Filter

9.12.6.26 PASSBAND FREQUENCY

Description: The maximum passband frequency is determined through the equation: $Passband_{user}$ = (Total Number of Points on Graph/Time Duration). The default value is 10% of the passband frequency.

9.12.6.27 STOPBAND FREQUENCY

Description: Stopband frequency is formulated; Stopband = $Passband_{user}$ + ($Passban_{max}$ - $Passband_{user}$). The default value depends on the $Passband_{user}$.

9.12.6.28 MAXIMUM PASSBAND RIPPLE (MAX PASSBRIPP)

Description: Passband ripple value is between 0.0 and 1.0 dB.

9.12.6.29 MINIMUM STOPBAND ATTENUATION (MIN.ATT.1.0+ DB)

Description: Stopband attenuation must be greater than 1.0 dB. The stopband frequency and maximum passband ripple should not be too close to either the given low- or high-end limits. Attenuation factors commonly range between 10 and 50. It is recommended that the user choose a passhand frequency that is approximately 5 to 20% of the number. Too small a number will cause a computational error and too large a number will cause the program to stall.

Usage: After the user enters the appropriate parameters, clicks the OK and accept the input parameters and exit the DATA CONTROL window.

Press Apply to display the filter result.

9.12.7 COMPUTATION

Description: This function allows the user to create a new curve through the algebraic operation of several selected curves including Add, Subtract, Multiply, Divide and so on.

ADD

This function enables two or more curves to be selected and added to each other.

SUBTRACT

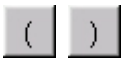
This function enables two or more curves to be selected and subtracted form each other.

MULTIPLY

This function enables two or more curves to be selected and multiplied.

DIVIDE

This function enables two or more curves to be selected and the values are divided to from a quotient curve.



BRACKET

This function enables the user to change the priority level of an expression.

TRANSLATE

This function enables the user to create a new curve with two selected curves. The new X-axis value is equal to the Y-axis value of the first selected curve and the new Y-axis value is equal to the Y-axis value of the second curve.

9.12.8 RESULT CURVE

Description: This function can display the expression dynamically during the operation. For example, if the user want to square a curve and integrate it. The user can click the curve name on the name list window, the name of the curve will display on the Result Curve text box. Then click the SQUARE button and the INTEGRATE button respectively. The content of the Result curve text box will change to such expression as **C27: Sq (): Int ()**. In this expression, the **C27** means the name of selected curve. **Sq ( )** and **Int ( )** mean the operation to this curve. There is a colon (:) following each operation.

There are two kinds of colour of the expression in the Result curve text box. One is black and another is red. Those two kinds colour denote two different kinds of meaning. When the colour of the expression is black, it means that the user can press the Apply to perform this operation. If the colour of the expression is red, it means that the expression is not correct or the condition of current operation is not satisfied so the operation can't be executed.

The user can input by keyboard to define the operation according to the format of the program. But this method is not very convenient for new user, so this method is not recommended for the new user.

9.12.8.1 APPLY

Description: This function allows the user to execute the operation after the user has selected some operation.

9.12.8.2 UNDO

Description: This function allows the user to cancel the last selected operation before pressing the Apply button.

9.12.8.3 CLEAR

Description: This function allows the user to cancel all the selected operations before pressing the Apply button.

NOTE: During the operation for curves, some curve data operations require the sample frequency and intervals are the same, e.g. binary arithmetic operation. Some other unary operations require a minimum number of samples e.g. Differentiation, Integration etc. If the user used these operations without respecting these requirements, an error message box will be popped up.

There are four digital filters as discussed earlier in Graph. All the filters have default parameters for convenience. There are two classes of digital filter, non-recursive and recursive. Butterworth filter is the lowpass recursive filter and the FIR and average filter are non-recursive filters. The computation requirement for the FIR filters is lower than that for Butterworth. However, the Butterworth gives more power and less signal loss.

9.12.9 NEW GRAPH

Description: This function allows the user to create the new graph on a new curve window and set as the current window. By default the new graph is plot on the current curve window if the user press Apply.

NOTE: Prior to opening multiple graph files, the user should open a NEW CURVE WINDOW for each graph. This will prevent the graphs from merging into a previously opened graph, unless it is needed.

9.12.10 SAVE

Description: This function allows the user to save all the graphs in current curve window.

9.12.11 EXIT

Description: This button allows the user to exit the Graph operation.

Chapter

10

Chapter 10: UTILITY

UTILITY includes the following sections: VIEWING OPTIONS, ICON BAR FUNCTIONS, PART CONTROL, UTILITY, and DISPLAY PARAMETER OPTIONS WINDOW. The functions in these sections allow the user to alter the model's display, define the model's parameters, and define the model's part makeup.

10.1 VIEW OPTIONS

The functions in the VIEWING OPTIONS menu are used to adjust the display of the items on the screen. See Figure 10.1.

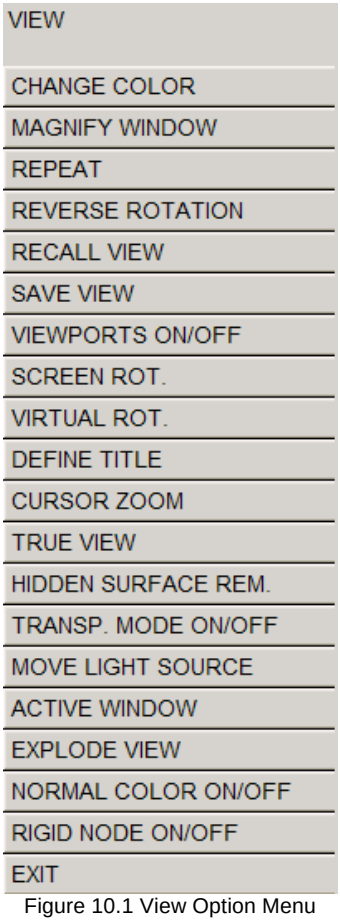


Figure 10.1 View Option Menu

A detailed description of each function is given in the following sections.

10.1.1 CHANGE COLORS

Description: The functions in this submenu allow the user to change the color of any displayed item (including part colors). See Figure 10.1.1.

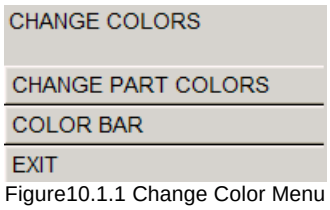


Figure10.1.1 Change Color Menu

- CHANGE PART COLORS

Description: This function changes the colors of selected parts. The user may activate a color change when the parts are on or off.

Usage:

1. VPG prompts:
   > SELECT PART TO CHANGE COLOR
   > PICK AN ELEM OR PART NAME OF A PART
   • The user selects a part as follows:
     A. By placing the cursor on a displayed part element and pressing the left mouse button.
     B. VPG displays the part list from which the user may select any part.

2. Once the user selects a part, VPG prompts:
   > SELECT COLOR FROM COLOR BAR OR EXIT
   • VPG displays the color bar on the right edge of the screen. The user may select a desired color via the mouse.

3. EXIT terminates this command.

- COLOR BAR

Description: Once the user executes this command, VPG displays the color bar and the assigned colors and numbers on the right edge of the screen. VPG removes the color bar when the user activates the REDRAW or CLEAR command.

10.1.2 MAGNIFY WINDOW

Description: This function magnifies an object in the display area.

Usage:

1. VPG prompts:
   > ENTER MAGNIFICATION
   • The user may enter any positive number as the magnification factor. If the number is greater than 1, it will cause the size of the object in the display area to be enlarged. If the number is less than 1, it will cause the size of the object in the display area to be reduced, e.g., an entry of 5 will enlarge the picture, whereas an entry of .5 will reduce the picture.

10.1.3 REPEAT

Description: This function is used in conjunction with the SCREEN AXES ROTATION and the VIRTUAL AXES ROTATION commands to repeat the degree of rotation and to replot the geometry of a part. The user may repeat this command as many times as desired.

10.1.4 REVERSE ROTATION

Description: This function works in conjunction with SCREEN AXES ROTATION and VIRTUAL AXES ROTATION to reverse the degree of rotation and to replot the geometry of a part. The user may repeat this command as many times as desired.

10.1.5 RECALL VIEW

Description: This function allows the user to recall any previously saved view. VPG contains the following predefined views.

1. TOP VIEW

2. **SIDE VIEW**
3. **REAR VIEW**
4. **ISOMETRIC VIEW (60 DEGREE ISOMETRIC)**

Usage:

When the user selects any of the above views, VPG re-scales the displayed part accordingly.

10.1.6 SAVE VIEW

Description: This function allows the user to save and store a desired view in the current VPG database.

Usage:

1. VPG prompts:
   > ENTER VIEW TITLE (UP TO 18 CHARACTERS)
   - The user enters a title for the desired view. RECALL VIEW lists the new view and its corresponding view number. The program allows the user to save up to six new views plus the four views that are predefined (a total of 10 views).

2. If the user exceeds the total number of views allowed, the prompt at the bottom of the screen will read:
   > SELECT VIEW TO OVERWRITE (5 to 10)
   - The user may select a view to overwrite and repeat step 1.

10.1.7 VIEWPORTS ON/OFF

Description: This function allows the user to view an object using the four predefined views available in the split-screen display mode.

Usage:

The user selects one of the following views: 1 - corresponds to the default view; 2 - corresponds to the TOP VIEW (X-Y PLANE) and the SIDE VIEW (X-Z PLANE); and 4 - corresponds to the TOP VIEW (X-YPLANE), the SIDE VIEW (X-ZPLANE), the ISOMETRIC VIEW and the REAR VIEW (Y-Z PLANE). Once the user executes the above option, the displayed model will automatically rescale itself to fill the split-screen display.

10.1.8 SCREEN AXES ROTATION

Description: This function allows the user to rotate an object in the display area according to an angle specified via the keyboard. The screen axes are fixed at all times and are defined as follows:

1. Screen X - axis is the direction from the left to right of the terminal screen.
2. Screen Y - axis is the direction from the bottom to the top of the terminal screen.
3. Screen Z - axis is the direction from the screen to the user.

Usage:

The user defines the rotation by entering the angle of a particular axis. The angle is defined as positive if the rotation corresponds to a counterclockwise direction (i.e., follows the right hand rule).

1. VPG prompts:

> ENTER 3 ANGLES TO ROTATE

- The user may now enter the desired X, Y, Z angles of rotation.

Note: The functions **REPEAT ROTATION** and **REVERSE ROTATION** are especially useful after executing the above command.

10.1.9 VIRTUAL AXES ROTATION

Description: This function allows the user to rotate an object about the global (or virtual) X-, Y-, and Z-axes according to an angle specified via the keyboard.

Usage:

1. VPG prompts:
   > ENTER 3 ANGLES TO ROTATE
   - The user enters the desired global X, Y, Z angles of rotation.

Note: The functions REPEAT ROTATION and REVERSE ROTATION are especially useful after executing the above command.

10.1.10 DEFINE TITLE

Description: This function allows the user to enter a title or a text label on the display screen.

Usage:

1. VPG prompts:
   > INPUT CHARACTER SIZE (1.0 - 10.0)

2. Once the user selects the text size, VPG prompts:
   > DEFINE TITLE STARTING LOCATION
   ? ABORT
   CURSOR LOCATION
   DONE
   EXIT
   UNDO LAST
   - The user places the text by positioning the cursor at the desired location and clicking the left mouse button.

3. VPG prompts:
   > ENTER TITLE
   - The user types in the text for the title and presses enter. The text appears at the previously selected location.

10.1.11 CURSOR ZOOM

Description: This function allows the user to zoom in on any displayed area in the Display Window by dynamically defining a window about a region of the screen.

Usage:

1. VPG prompts:
   > DEFINE ZOOM WINDOW
   - The user defines the upper left corner of the zoom window by positioning the cursor on the display screen. The user then presses the left mouse button and drags the cursor diagonally down the screen until the desired window size is reached. The user clicks the mouse once to activate the window

(highlighted in white).

10.1.12 TRUE VIEW

Description: This function allows the user to display an object in true view, or the normal view of the local W-axis as projected onto the local UV plane.

Usage:

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/POINT FOR ORIGIN
   • VPG asks for a desired coordinate system (local or global).

2. VPG displays the coordinates on screen and prompts:
   > ACCEPT. (Y/N/A)
   • NO returns the user to step 1.
   • YES continues this command to step 3.
   • ABORT

3. VPG prompts:
   > KEEP ANY GLOBAL AXIS VERTICAL. (Y/N)
   ? YES
   > ENTER THE VERTICAL GLOBAL AXIS (x,y,z)
   • The user may select an axis to be displayed from the bottom to the top of the screen. The object remains in its true view if this option is selected.
   No
   • NO displays the object in its true view (perpendicular to the W-axis). This option displays the desired geometry in its true view according to the global X, Y, and Z coordinates or to the Z-axis (displayed vertically).

4. VPG prompts:
   > DO YOU WISH TO MAP ANY LINE ON TO THE X-Y PLANE.
   ? YES
   NO exits the TRUE VIEW function

5. If the user selects YES, VPG prompts:
   > SELECT LINES
   ? ABORT
   EXCLUDE ON/OFF
   LINE
   MULTI-POINT REGION
   PART
   WINDOW
   REJECT LAST
   DONE

6. Selecting DONE copies the selected lines along the XY plane in true view.

10.1.13 HIDDEN SURFACE REMOVAL (toggle)

Description: This function improves the integrity of the 3D simulation of a model. The user may toggle the hidden lines on and off while using the SHADING and FILL HIDE PLOT commands.
This effect creates opaque elements that prevent objects in the background from showing through objects in the foreground.

Usage:

1. VPG prompts:
   > HIDDEN SURFACE REMOVAL ON or
   > HIDDEN SURFACE REMOVAL OFF

Note: When checking for penetration, make sure the flag HIDDEN SURFACE REMOVAL=ON appears along the bottom of the display area.

10.1.14 TRANSPARENCY MODE ON/OFF (toggle)

Description: This function toggles the transparency mode on or off during shading. Transparency factors are defined in the SETUP menu.

Usage:

1. VPG prompts:
   > TRANSPARENCY MODE ON or
   > TRANSPARENCY MODE OFF

10.1.15 MOVE LIGHT SOURCE

Description: VPG uses two types of light sources: ambient light and directional light.

Ambient light comes from all directions and illuminates objects regardless of their position or orientation with the same intensity.

Directional light comes from a specific direction. It flows uniformly and maintains a consistent intensity. It has a directional vector that determines the face on which an object receives light.

The MOVE LIGHT SOURCE command positions a directional vector at two points that are perpendicular to an object. The right and left arrow keys move the light source in the positive and negative X-direction, while the up and down arrow keys move the light source in the positive and negative Y-direction.

10.1.16 ACTIVE WINDOW

Description: This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. The user defines the region by dragging a window over the desired area， by specifying the volume of space via the keyboard with X,Y,Z coordinates, by two diagonal points/nodes, by multiple point region or by group elements. VPG displays the elements, lines, and surfaces within the window (volume in space) as active. Other objects on screen remain masked and inactive. See Figure 10.1.16

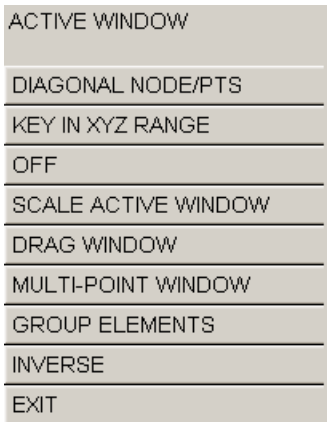


Figure 10.1.16 Active Window Menu

Note: An ACTIVE WINDOW may be defined at any time within any menu of VPG by clicking the right mouse button.

Usage:

1. VPG prompts:
   > SELECT OPTION FOR ACTIVE WINDOW
   ? DIAGONAL NODE/PTS
   KEY IN XYZ RANGE
   ACTIVE WINDOW OFF
   SCALE ACTIVE WINDOW
   DRAG WINDOW
   MULTI-POINT WINDOW
   GROUP ELEMENTS
   INVERSE
   EXIT

-DIAGONAL NODE/POINTS

The user selects two nodes/points in a diagonal fashion to define the active widow.

-KEY IN XYZ RANGE

The user defines the size (volume) of the active window by entering its global coordinates via the keyboard. The user may also expand an existing active window by modifying any or all of its coordinates.

> ENTER XYZ RANGE (XMIN, XMAX, YMIN, YMAX, ZMIN, ZMAX)

-ACTIVE WINDOW IS OFF

Turns off or deactivates the current active window.

-SCALE ACTIVE WINDOW

This allows the user to scale the current active window to a larger or smaller window.

> ENTER SCALE FACTOR

-DRAG WINDOW

The user defines the active window by selecting a two-region diagonal on screen.

-MULTI-POINT WINDOW

The user defines an active multi-point window by cursor, window starts with left button, and middle button will close and complete the window.

-GROUP ELEMENTS

VPG prompts user to select elements, the selected elements will be active, and others will be inactive.

-INVERSE

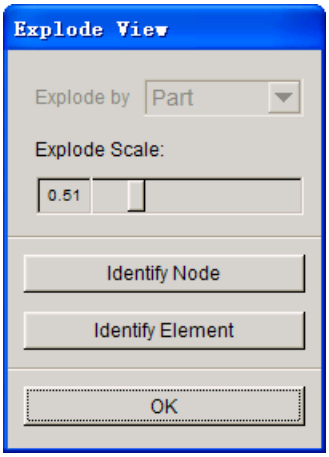
Active models including geometry/elements will be inactive, those inactive will change to be active.

10.1.17 EXPLODE VIEW

Description: This function shows an explode view for the model, user can identify node and elements under explode surroundings.

Usage:

1. VPG pops up a dialog window:



- **Explode by** user can select by part or part set.
- **Explode Scale** set a explode scale level, default is 0.00
- **Identify Node** identify a node under explode surroundings.
- **Identify Element** identify an element under explode surroundings.
- **OK** exit and terminate explode effect.

10.1.18 NORMAL COLOR ON/OFF

Description: This function toggles normal color mode on or off for showing element normal or orientation.

Usage:

1. VPG prompts:
   > NORMAL/ORIENTATION COLOR ON or
   > NORMAL/ORIENTATION COLOR OFF

- When this option is on, the normal or orientation will be shown with colour instead of arrow.

10.1.19 RIGID NODE ON/OFF

Description: This function toggles the rigid body node on or off when creating rigid link.

Usage:

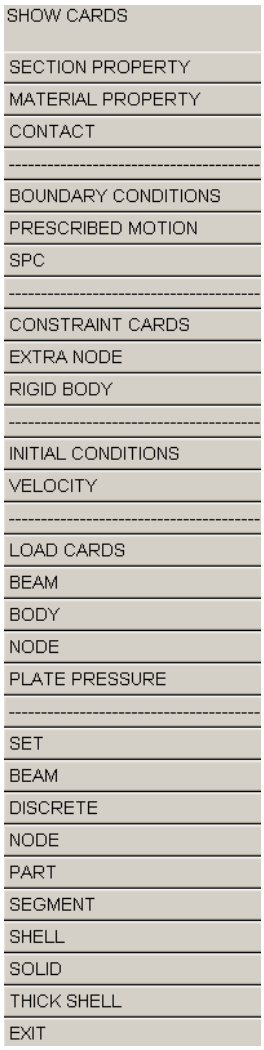
1. VPG prompts:
   > RIGID BODY NODE FILTER IS ON or
   > RIGID BODY NODE FILTER IS OFF
   - When this option is on, rigid body node will be shown as highlighted spots when try to create rigid link.

10.1.20 ROTATE CENTER ON/OFF

Description: This function helps to rotate model with a specified center. When this option is on, and user clicks the icon Free Rotation, VPG will prompt user to select a rotation center.

10.1.21 SHOW CARDS

Description: This function integrates all showing functions in VPG into a total menu page, user here can show material, section, contact, boundary condition, initial condition, constraint, load and set, please refer following figure, and user can also find these functions in other menus of VPG.



10.2 ICON BAR

The ICON BAR contains commonly used commands.

Database and Part control

Viewing manipulation

The user initiates the commands via mouse pick. A detailed description of each function is given in the following section.

10.2.1 DATABASE AND PART CONTROL

NEW FILE

Allows the user to create a new database file. For a detailed description, see chapter 5 FILE MANAGER.

OPEN FILE

Allows the user to open a database file. For a detailed description, see chapter 5 FILE MANAGER.

SAVE FILE

Allows the user to save a database file. For a detailed description, see chapter 5 FILE MANAGER.

PRINT

Allows the user to create a postscript file of display area and send the file to the printer (default) or to a file. Prior to printing, the postscript driver must be initialized to accommodate the eta/VPG software.

Note: The user can change the Printing Defaults in the SETUP/HARDCOPY DEFAULT menu.

PART ON/OFF

Allows the user to turn selected parts on or off. The PART TURN ON/OFF dialog window is displayed once the icon is selected. For a description of the command PART ON/OFF, refer to PARTS/PART ON|OFF in Section 10.3.

10.2.2 VIEWING MANIPULATION

SX - SCREEN X ROTATI*ON*

This shortcut rotates the model about the screen X-axis, where the X-axis is parallel to the horizontal edge of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

SY - SCREEN Y ROTATION

This shortcut rotates the model about the screen Y-axis, where the Y-axis is parallel to the vertical edge of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

SZ - SCREEN Z ROTATION

This command rotates the model about the screen Z-axis, where the Z-axis is perpendicular to the plane of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate;

tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VX - VIRTUAL X ROTATION

This command rotates the model about the global X (the X-axis is as displayed in the coordinate system at the lower left side of the screen). When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VY - VIRTUAL Y ROTATION

This command rotates the model about the global Y (the Y-axis is as displayed in the coordinate system in the lower left hand corner of the screen). When the user selects this command, VPG displays a ruler bar on the right hand side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VZ - VIRTUAL Z ROTATION

This command rotates the model about the global Z (the Z-axis is as displayed in the coordinate system in the lower left corner of the screen). When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

TOP VIEW

This command automatically displays the model from the TOP or XY-plane.

SIDE VIEW

This command automatically displays the model from the SIDE or XZ-plane.

REAR VIEW

This command automatically displays the model from the REAR or YZ-plane.

ISOMETRIC VIEW

This command automatically displays the model form the ISOMETRIC (60 Degree Isometric) point of view.

FREE ROTATION

The Freehand Rotation (FREROT) command is a combination of SX and SY. Moving the mouse up/down manipulates SX. Moving the mouse left/right manipulates SY. Moving the mouse diagonally combines the movements of both commands. Clicking the left mouse button stops the rotation. Pressing Control and the left mouse button can also activate this function.

PAN

This command translates the model by following the movement of the cursor. If the cursor is moved off the screen, the cursor reappears at the center of the screen. Clicking the left mouse button stops the pan. Pressing Control and the middle mouse button can also activate this function.

CURSOR ZOOM

The user picks a point about which to zoom. The model is centered about this point and the User may move the cursor up or down to zoom in and out. Pressing Control and the right mouse button can also activate this function.

WINDOW ZOOM

The user defines the corners of the zoom window by positioning the cursor on the display screen. The user presses the left mouse button and drags the mouse diagonally down until the desired window size is reached. Release the left button. The section included in the window is displayed in full screen.

FREE HAND ZOOM

The user defines the area of zoom window by clicking and holding the left mouse button in the display area and drawing a circle around the interested area. When the left button is released, the area that had been circled is displayed full-screen.

FILL

This command rescales the model to include all entities belonging to parts that are currently turned on. FILL automatically zooms in or out until the model fits the viewing area of the screen.

ACTIVE WINDOW

This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. For a detailed description, see section 10.1.16 VIEW/ACTIVE WINDOW.

REDRAW

VPG is currently designed to update the screen after each command. Occasionally, the use of specific commands requires that images be updated with an additional step
(e.g., when the user operates the dynamic zoom at the same time that the element normals are displayed, he or she must activate the REDRAW command to resize the arrows that represent the element normals).

CLEAR

Allows the user to remove the highlighted entities from the screen such as those that are made with the commands SHOW LINE, BOUNDARY CHECK, ID ELEMENT, DEFINE TITLE, etc.

CLEAR NODE

Clear the display of connecting nodes

CLEAR ELEMENT

Clear the display of connecting elements

IDENTIFY NODE

This function enables the user to identify any node.

IDENDFIY ELEMENT

This function enables the user to identify any element.

DISTANCE BETWEEN TWO NODES

This function enables the user to calculate the distance between two nodes.

ANGLE BETWEEN THREE NODES

This function enables the user to measure the angle between two vectors formed by three nodes.

COORDINATES ON/OFF

This function enables the user to turn on/off coordinate systems.

10.3 PART CONTROL

The functions of the PART CONTROL menu are intended for organizing line, surface, and element data in a structure. A PART is a set of lines, surfaces, and/or elements grouped under a PART NAME. The PART NAME is defined as a label of alphanumeric data that consists of no more than eight characters. Presently up to 1000 parts may be defined in a database. Each part has a unique Part Identification (PID). The functions in the PART CONTROL menu are organized as in Figure 10.3.1:

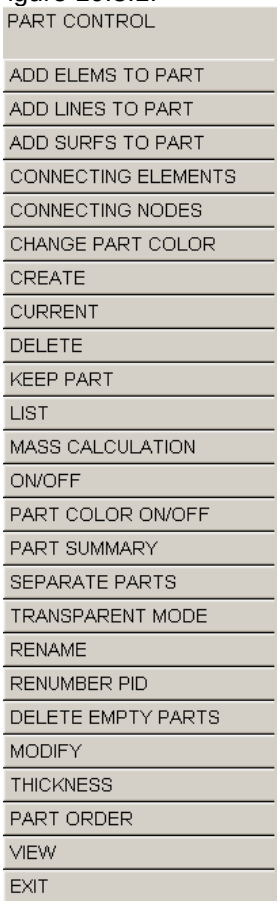


Figure10.3.1 Part Control Menu

A detailed description of each function is given in the following sections.

10.3.1 ADD ELEMENTS TO PART

Description: This function allows the user to copy elements from one part to another part. The part to which the elements are added does not have to be current or on.

Usage:

1. VPG prompts:
   > SELECT TARGET PART
   > PICK AN ELEM OR PART NAME OF A PART

2. Once the user selects the target part, VPG prompts:
   > ELEMENTS WILL BE ADDED TO PART xxxx [part name].

> SELECT ELEMENTS

3. The user selects the desired elements, then enters DONE to exit the menu.

10.3.2 ADD LINES TO PART

Description: This function allows the user to copy lines from one part to another part. The part to which the lines are added does not have to be current or on.

Usage:

1. VPG prompts:
   > SELECT TARGET PART
   > PICK AN ELEM OR PART NAME OF A PART

2. Once the user selects the target part, VPG prompts:
   > LINES WILL BE ADDED TO PART xxxx (part name).
   > SELECT LINES

3. The user selects the desired lines, then enters DONE to exit the menu.

10.3.3 ADD SURFACES TO PART

Description: This function allows the user to copy surfaces from one part to another part. The part to which the surfaces are added does not have to be current or on.

Usage:

1. VPG prompts:
   > SELECT TARGET PART
   > PICK AN ELEM OR PART NAME OF A PART

2. Once the user selects the target parts, VPG prompts:
   > SURFACES WILL BE ADDED TO PART xxxx [part name].
   > SELECT SURFACES

3. The user selects the desired surfaces, then enters DONE to exit the menu.

10.3.4 CONNECTING ELEMENTS

Description: This function is used to determine element connectivity from desired part(s) in the model to others.

Usage:

1. VPG prompts:
   > SELECT PART FOR CONNECTING ELEMENTS
   > PICK AN ELEMENT OR A PART NAME OF A PART

2. After selecting the desired part(s), VPG prompts:
   > SLECTED PART ID: XX; NAME XXXXX
   > XXXX ELEMENTS FROM XXX PARTS CONNECTED TO XXXX
   > PICK AN ELEMENT OR PART NAME TO TURN ON
   - The connecting elements are highlighted. User can select NEIGHBOR NEXT to show more elements to displayed shells.

3. Once the part is selected, VPG prompts:
   > PART XXXX IS TURNED ON
   > PICK AN ELEMENT OR PART NAME TO TURN ON
   • This step is repeated until the user selects exit.

10.3.5 CONNECTING NODES

Description: This function is used to determine node connectivity from one part in the model to another.

Usage:

1. VPG prompts:
   > SELECT PART FOR CONNECTING ELEMENTS
   > PICK AN ELEMENT OR A PART NAME OF A PART
   <part list>
   CURSOR AT ELEMENT
   CURSOR AT LINE
   CURSOR AT SURFACE
   CURRENT PART
   UNDO EXIT

2. After selecting the desired part, VPG prompts:
   > PART XXXXX IS SELECTED
   > XXXX ELEMENTS FROM XXX PARTS CONNECTED TO XXXX
   > PICK AN ELEMENT, OR PART NAME TO TURN ON
   <connected part list>
   ALL CONNECTED PARTS
   SELECT BY ELEMENT
   UNDO LAST EXIT
   • The connecting nodes are highlighted.

3. Once the part is selected, VPG prompts:
   > PART XXXX IS TURNED ON
   > PICK AN ELEMENT OR PART NAME TO TURN ON
   • This step is repeated until the user selects exit.

10.3.6 CHANGE PART COLORS

Description: This function allows the user to change the color of parts.

Usage:

1. VPG prompts:
   > SELECT PART TO CHANGE COLOR
   > PICK AN ELEM OR PART NAME OF A PART

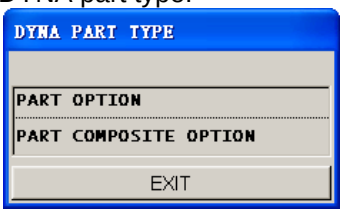
2. Once the user selects target part(s), VPG prompts:
   > SELECT COLOR FROM COLOR BAR OR EXIT
   • The user selects the desired color from the color bar.

10.3.7 CREATE

Description: This function creates a new part in the VPG database. The user must name the new part with any combination of up to eight alphanumeric characters.

Usage:

1. VPG displays following DYNA part type:



• If **PART OPTION** is selected, go to step 2, if PART COMPOSITE OPTION is selected, go to step 3.

2. If **PART OPTION** is selected, VPG displays the PART DEFINITION window as below. See figure 10.3.2.

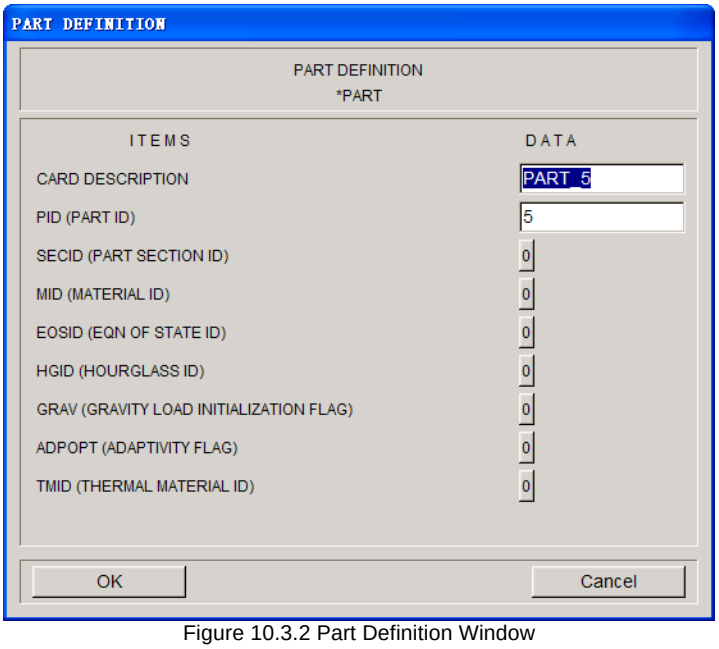
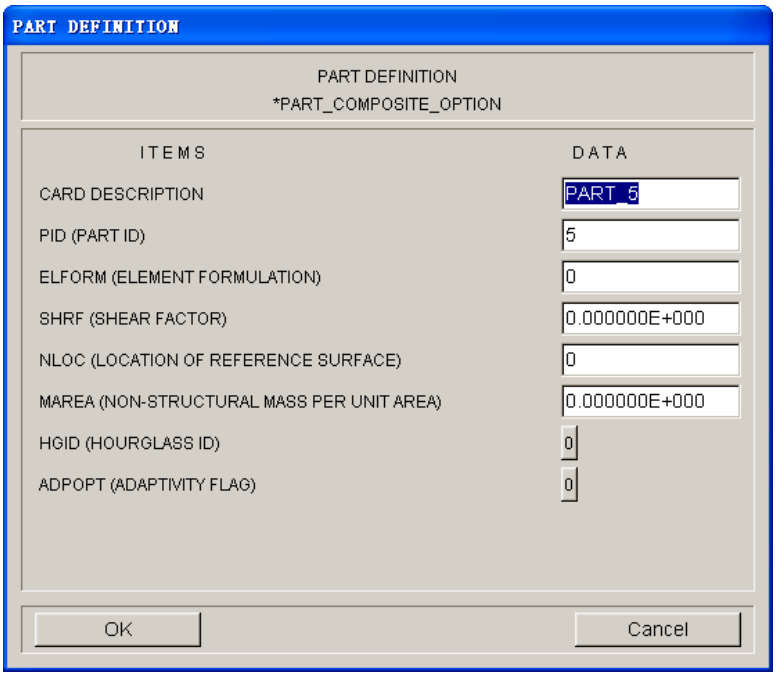


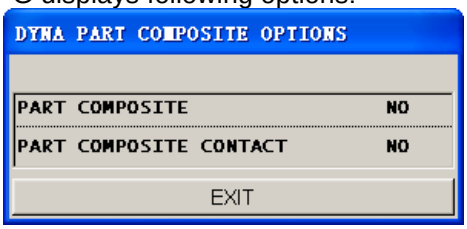
Figure 10.3.2 Part Definition Window

• The user edits the window to create the part. If SECID, MID, EOSID, HGID, or TMID is selected, the user is prompted to edit another window for these properties before retuning to the PART DEFINITION window. OK completes the part and returns the user to the PART CONTROL menu. The materials and properties of a new part do not have to be defined at the time the part is created.

3. If **PART COMPOSITE OPTION** is selected, VPG displays the PART DEFINITION window as below.



4. After clicking OK, VPG displays following options:



- The user can select these two options to define *PART_COMPOSITE or *PART_COMPOSITE_CONTACT. Please refer to DYNA manual for reference.

Note: The new part is displayed as current.

10.3.8 CURRENT

Description: This function changes the current part. All lines, surfaces, and elements that the user creates are automatically included in the current part. The name of the current part is displayed in its part color in the lower right corner of the screen.

Usage:

1. VPG prompts:
   > SELECT NEW CURRENT PART
   > PICK A LINE (L), ELEM (C), OR PART NAME (P) OF A PART.
   - The user selects a target part to make current.
   - Once the user selects the target part, VPG makes the part current and displays the part name in the lower right hand corner of the screen. The user can also access the CURRENT PART function by clicking on the current part name displayed in the DISPLAY PARAMETER OPTIONS WINDOW.

10.3.9 DELETE

Description: This function allows the user to delete a part with its associated surfaces, lines, and elements. The nodes of a part that are selected for deletion become unreferenced and appear as white asterisks ("*") on the screen.

Note: If deleting the CURRENT PART, the user must select a new current part in order to continue model generation.

Usage:

1. VPG prompts:
   > PICK A LINE (L), AN ELEM (C), OR PART NAME (P) OF A PART
   - The user selects a target part.

   > XXXX LINES, XXXX SURFACES AND XXXX ELEMENTS IN PART YYYY.

2. VPG prompts:
   > SELECT DELETE SELECTED PART OPTION
   PARTS ONLY
   PART/NODES
   ABORT
   - The user selects a deletion option. PARTS ONLY deletes the part(s) selected along with any element definition and geometric entities. All nodes will remain as UNREFERENCED NODES. PARTS/NODES will delete the UNREFERENCED NODES in addition to element and geometry data. ABORT will return the user to the PART MENU

3. Once the user selects the option, VPG prompts:
   > XXXX LINES, XXXX SURFACES AND XXXX ELEMENTS IN PART YYYY.
   > O.K. TO DELETE PART YYYY.
   ? YES
   NO
   - YES deletes the selected part and its lines, surfaces, and elements.
   - NO deselects the part.

10.3.10 KEEP PART

Description: This function retains the ON status of the selected part(s), and turns OFF the deselected parts. The program automatically re-scales the selected part(s) to fill the screen.

Usage:

1. VPG prompts:
   > SELECT PART TO KEEP
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   - The user selects a target part.
   - VPG highlights the selected line(s) or element(s).

2. Once the user selects the desired part(s), VPG prompts:
   > PART xxxx (part name) IS SELECTED

3. EXIT terminates this function after replotting and rescaling the selected parts.

10.3.11 LIST

Description: This function lists the part names with the corresponding PID numbers and

ON/OFF status. Parts that are currently ON are listed in their part color. Parts that are currently OFF are listed in white.

10.3.12 MASS CALCULATION

Description: This function determines the mass of a part, a group of parts, or the entire model.

Usage:

1. VPG prompts:
   > ENTER OUTPUT FILE NAME OR [CR] FOR DEFAULT OR "STOP" FOR QUIT

2. Once the file name is entered VPG prompts:
   > ENTER COMMAND
   ALL PARTS
   SELECT PART
   EXIT

3. VPG displays the SUMMARY OF MASCAL window with the part(s) properties and displays in the dialogue window the part(s) density.

10.3.13 ON/OFF

Description: This function toggles selected parts ON and OFF, and the parts listed can be sorted by Part ID or Part Name with the triangle icon in the part list window. The following options are given as a submenu. See Figure 10.3.3.

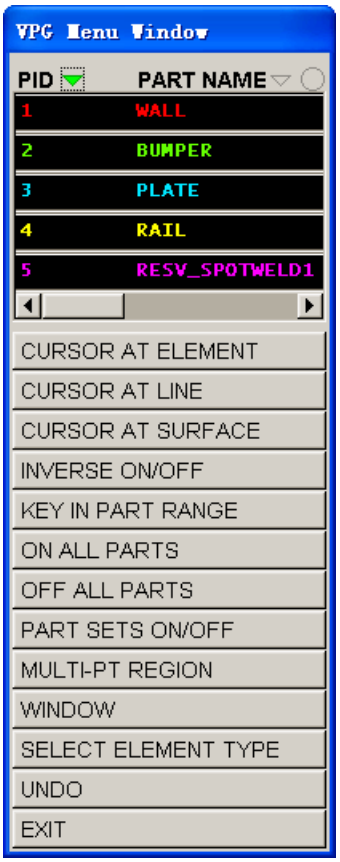


Figure 10.3.3 Part On/Off Menu

Usage:

1. VPG prompts:
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   PART NAME - clicking on the listed parts will turn them on/off
   CURSOR AT ELEMENT -selects part by element - **DEFAULT**
   CURSOR AT LINE -selects part by line
   CURSOR AT SURFACE -selects part by surface
   INVERSE ON/OFF -all on parts off and all off parts on
   KEY IN PART RANGE - selects by assigning a part number range
   ON ALL PARTS -all parts are turned on
   OFF ALL PARTS -all parts are turned off
   PART SETS ON/OFF -turns parts on/off by their assigned set
   MULTI PT REGION -selects part by user defined multi point region
   WINDOW -selects part by drag window
   SELECT ELEMENT TYPE -selects element type
   UNDO -rejects last

10.3.14 PART COLORS ON/OFF (toggle)

Description: This function changes each element type into a separate and uniform color (i.e., quadrilateral elements turn blue, triangular elements turn red, 8-node cube elements turn green, 6-node wedge elements turn magenta, and 4-node tetrahedron elements turn light magenta). This function activates the color changes when selected once, and restores the original colors when selected again.

10.3.15 PART SUMMARY

Description: This function displays statistics regarding the geometry, model, material, and interface information of the selected parts. The figures are displayed in the options area.

Usage:

1. VPG prompts:
   > SELECT PART FOR SUMMARY
   > PICK AN ELEMENT, OR PART NAME OF A PART.

   Note: DRAG WINDOW and MULTI-POINT REGION are not valid selections for this command.

2. The following figure is an example of the statistics display for selected parts:

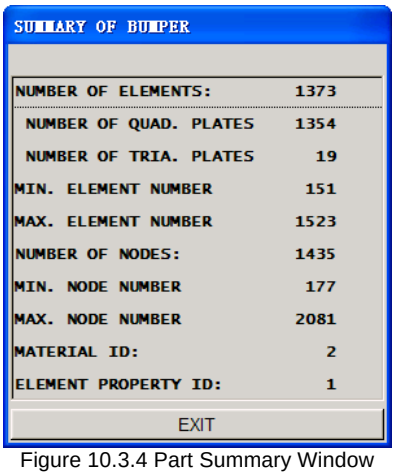


Figure 10.3.4 Part Summary Window

10.3.16 SEPARATE PART(S)

Description: This function allows the user to separate part(s) from a structure. For example, two parts that are connected by common nodes may be quickly separated (or disconnected) using SEPARATE PART. This function also provides the user with the option to create zero length RBARS between the separated parts. (Common nodes between parts become new nodes once the parts are separated.)

Usage:

1. VPG prompts:
   > CREATE RIGID LINKS BETWEEN SEPARATED PARTS. (Y/N)
   - YES creates a part called ZEROBAR, in PART CONTROL, that automatically contains all of the newly-created RBARS.

2. VPG prompts:
   > SELECT PART OR ALL. (SP/A/E)

3. If the user selects ALL PARTS, VPG prompts:
   > XXXX (# of RBE2s) RBE2S CREATED BETWEEN SEPARATED PARTS
   All modeled parts in the database separate from each other. This includes:
   PLATES
   SOLIDS
   CBARS/CBEAMS
   CELAS1/CELAS2 ELEMENTS
   PLOTEL ELEMENTS
   CROD/CTUBE ELEMENTS
   These entities do not separate:
   RBE2/RBAR ELEMENTS
   CONM2 ELEMENTS
   CORD2R ELEMENTS

4. If the user enters SELECTED PARTS, VPG prompts:
   > SELECT PART TO SEPARATE
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   - The parts separate from each other, and VPG prompts:

   > XXXX (# of RBE2s) RBE2S CREATED BETWEEN SELECTED PARTS.

10.3.17 TRANSPARENT MODE

Description: This function makes selected parts transparent during the shading operation.

Usage:

1. VPG prompts:
   > PICK A LINE (L), ELEM (C), OR PART NAME (P) OF A PART

2. The user makes the appropriate selection, then enters DONE to exit the menu.

Note: The user may adjust the degree of transparency in the SETUP menu (refer to Section 10.4).

10.3.18 RENAME

Description: Allows the user to change a selected part's name.

Usage:

1. VPG prompts:
   >SELECT PART TO RENAME
   >PICK AN ELEMENT OR PART NAME OF A PART

2. After the user selects a part, VPG prompts:
   >ENTER NEW PART NAME OR EX TO EXIT

10.3.19 RENUMBER PID

Description: This function allows the user to renumber all parts' PID in the existing database. See Figure 10.3.5.

Usage:

1. VPG prompts:
   > SELECT OPTION

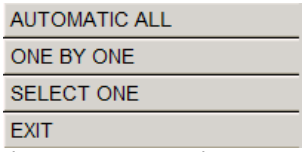


Figure 10.3.5 Renumber Part ID

10.3.20 DELETE EMPTY PARTS

Description: This function allows the user to delete parts that contain no lines, surfaces, or elements.

10.3.21 MODIFY

Description: Allows the user to change a selected part's attributes such as thickness, material, etc.

Usage:

1. VPG prompts:
   > PICK ELEMENT OR PART NAME OF A PART

2. After the user selects a part, a Part Definition window appears.

10.3.22 THICKNESS

Description: This function allows the user to calculate and apply a nodal thickness value for each part of the model.

The calculation of the thickness is made via a “one-step” metal forming analysis. As the part is selected, a background process calculates the nodal thickness values based on material properties, the shape of the component, and the initial component thickness.

Note: This process is available only on HP-UX operating systems and requires an additional license key.

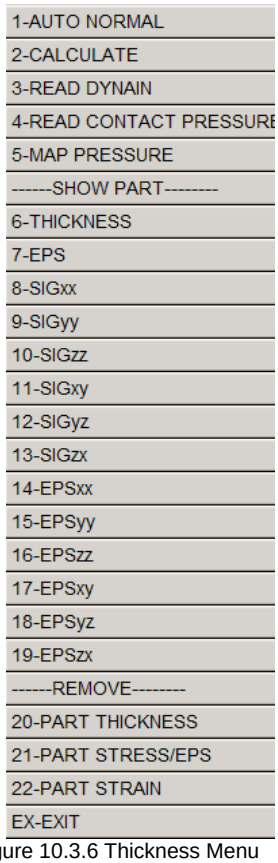


Figure 10.3.6 Thickness Menu

-AUTO NORMAL

Description: The calculation method used to create nodal thicknesses requires that all elements be consistent with respect to normals. This requires element connectivities to be defined in a consistent manner. This function allows users to check the connectivity definitions for the components and reverse any elements that are inconsistent.

Usage:

1. VPG prompts:
   > SELECT PART FOR AUTO NORMAL
   ? ALL ACTIVE PARTS
   CURSOR PICK PART

2. After the user selects the desired parts, VPG prompts:
   > IS NORMAL DIRECTION ACCEPTABLE? (Y/N)

-CALCULATE

Description: This function executes a background process that uses component geometry, material, and thickness to calculate a nodal thickness and residual strain value.

-READ DYNAIN

Description: This function allows users to read a DYNAIN file from a previously executed LS-DYNA metal forming analysis.

Usage:

1. VPG prompts:
   > INPUT DYNAIN FILE NAME

-READ CONTACT PRESSURE

Description: This function allows users to read a pressure file from a previously executed LS-DYNA metal forming analysis.

Usage:

1. VPG prompts:
   > INPUT PRESSURE FILE NAME EXPORTED BY ETA-POST

-MAP PRESSURE

Description: This function allows users to map pressure from a part to another part

-SHOW THICKNESS

Description: This function allows the user to visualize the calculated thickness values via color fringe plots. The thickness value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPS

Description: This function allows the user to visualize the calculated resultant strain values via color fringe plots. The resultant strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGxx

Description: This function allows the user to visualize the calculated xx component stress values via color fringe plots. The xx component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGyy

Description: This function allows the user to visualize the calculated yy component stress values via color fringe plots. The yy component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGzz

Description: This function allows the user to visualize the calculated zz component stress values via color fringe plots. The zz component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGxy

Description: This function allows the user to visualize the calculated xy component stress values via color fringe plots. The xy component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGyz

Description: This function allows the user to visualize the calculated yz component stress values via color fringe plots. The yz component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW SIGzx

Description: This function allows the user to visualize the calculated zx component stress values via color fringe plots. The zx component stress value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSxx

Description: This function allows the user to visualize the calculated xx component strain values via color fringe plots. The xx component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSyy

Description: This function allows the user to visualize the calculated yy component strain values via color fringe plots. The yy component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSzz

Description: This function allows the user to visualize the calculated zz component strain values via color fringe plots. The zz component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSxy

Description: This function allows the user to visualize the calculated xy component strain values via color fringe plots. The xy component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSyz

Description: This function allows the user to visualize the calculated yz component strain values via color fringe plots. The yz component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-SHOW EPSzx

Description: This function allows the user to visualize the calculated zx component strain values via color fringe plots. The zx component strain value shown on the fringe plot will correspond to the color bar and legend shown on the right side of the display area.

-REMOVE THICKNESS

Description: This function allows the user to remove previously defined thickness values obtained from a thickness calculation. The user must then recreate a shell section or material thickness using the appropriate modeling parameter.

-REMOVE STRESS/EPS

Description: This function allows the user to remove previously defined stress and resultant strain values obtained from calculation.

-REMOVE STRAIN

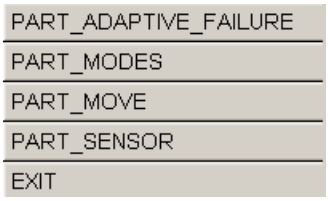
Description: This function allows the user to remove previously defined strain values obtained from a thickness calculation.

10.3.23 VIEW

Description: The functions in the VIEW menu are used to adjust the display of the items on the screen. See section 10.1.

10.3.24 SPECIAL PART KEYWORD

Description: The functions in this menu are used to define special keywords about PART. Those keywords are as following:



All options use following functions:

CREATE: create a special part keyword card
DELETE: delete already created special part keyword cards
LIST: shows a list already created special part keyword cards
MODIFY: prompts the user to select a special part keyword card to modify

10.4 UTILITY MENU

The commands in the UTILITY menu make up VPG's "tool kit." Many of these commands are also located in other menus as well.

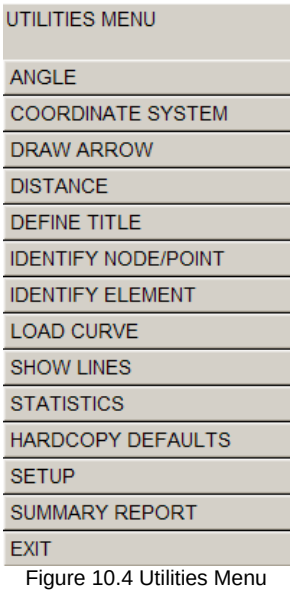


Figure 10.4 Utilities Menu

A detailed description of each function is given in the following sections

10.4.1 ANGLE

Description: The ANGLE command measures the angle between three selected points and/or nodes. VPG calculates the measurements of four angles for this command. The first is the "true" angle -- the angle in the plane of the selected points. The remaining three angles are measured from the TOP, SIDE, and REAR views of the model.

Usage:

1. VPG prompts:
   > SELECT VERTEX

2. VPG prompts:
   > SELECT 2 POINTS/NODES

3. VPG displays angle information:
   > ANGLE= XX.XX, TOP: XX.XX, SIDE: XX.XX, REAR: XX.XX

4. DONE or EXIT returns the user to the UTILITY menu.

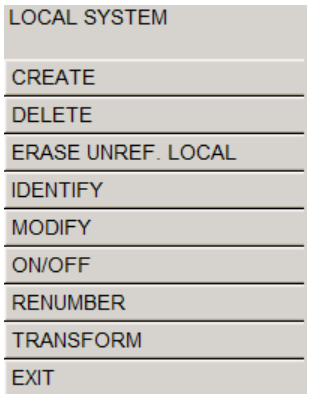
10.4.2 COORDINATE SYSTEM

Description: The functions in this menu create and modify local coordinate systems.

Note: The COORDINATE SYSTEM command requires several submenus. To help guide the user, each submenu is described in the following subsections.

Usage:

1. VPG prompts:
   > ENTER COMMAND



10.4.2 Coordinate System Menu

2. After the user enters the appropriate system, VPG lists the options specific to that system.

-CREATE

Description: This function creates local coordinate systems. The CS command is used in conjunction with the ASSIGN COORDINATE SYSTEM TO NODE command.

Usage:

1. VPG prompts:
   > SELECT LOCAL SYSTEM TYPE
   NODES
   SYSTEM
   VECTOR

2. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Local Coordinate System, Section 2.16.

3. After the user enters one of the options above, the local coordinate system appears on screen with its C.S. number.

-DELETE

Description: This function deletes local coordinate system sets from the database.

Usage:

1. VPG displays the DEFINE COORDINATE CARD LIST and prompts:
   > SELECT CARD TO DELETE

2. EXIT deletes the selected coordinate system from the database and resets the nodes to the global system.

Note: To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the global coordinate system (refer to ASSIGN COORDINATE SYSTEM on page 3 of this section).

-ERASE UNREF. LOCAL

Description: This function allows the user to delete any local coordinate systems not referenced by any entity in the model.

-MODIFY

Description: This function modifies the definition of an existing local coordinate system.

Usage:

1. VPG displays the define coordinate card list and prompts:
   > EXIT TO CONTINUE

2. After selecting the coordinate system, VPG displays the CSYS DEFINITION CARD, which allows the user to select and edit the coordinate systems data.

3. Once the user accepts the modified card, the new local system appears on the screen. All local systems belonging to the modified coordinate system automatically adjust to match the labeled coordinate system.

-CURRENT

Description: This function switches the current coordinate system from local to global.

-DISPLAY ON/OFF

Description: This function toggles the display of local coordinate systems on or off.

-RENUMBER

Description: This function changes the ID of selected local coordinate systems. The user may select individual local coordinates, or may renumber all, or a portion of the local coordinate systems in the model database.

-TRANSFORM

Description: This function allows the user to change the origin of coordinate systems.

10.4.3 DRAW ARROW

Description: This function allows the user to draw arrows about a specific image on the display screen. Arrows created using this function may be removed from the display by selecting the CLEAR function from the icon bar.

Usage:

1. VPG prompts:
   > PICK THE TAIL LOCATION FOR THE ARROW
   ? CURSOR LOCATION
   EXIT
   UNDO LAST

2. After the user selects the location of the tail, VPG prompts for the location of the head.
   > PICK THE HEAD LOCATION FOR THE ARROW
   ? CURSOR LOCATION
   EXIT
   UNDO LAST

3. After the user defines the location of the head, VPG draws the arrow and returns to

step 1. See figure 10.4.3.

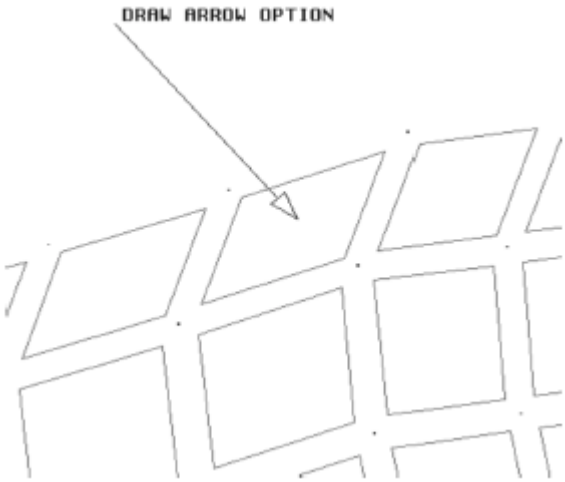


Figure10.4.3 Draw Arrow

10.4.4 DISTANCE

Description: This function calculates the distance between 2 points, 2 nodes, or a node and a point. This command is also listed in the **ELEMENT OPTIONS** and **NODE OPTIONS** menus.

Usage:

1. VPG prompts the user to select a node or point:
   > SELECT TWO POINTS/NODES
   - After the user selects the appropriate points or nodes, VPG displays in the prompt area of the screen the vector distance and the DX, DY, and DZ components.

2. ABORT, DONE, or EXIT terminates this function.

10.4.5 DEFINE TITLE

Description: This function allows the user to enter a title or text label at any location on the display screen. The title may be removed by selecting the CLEAR function from the icon bar.

Usage:

1. VPG prompts:
   > INPUT CHARACTER SIZE (1.0 - 10.0) <1.0>

2. After the user selects a font size, VPG prompts:
   > DEFINE TITLE STARTING LOCATION
   - The user defines the location of the text by positioning the cursor at the desired point on the screen and clicking the left mouse button.

3. VPG prompts:
   > ENTER TITLE
   - The user types in the text and presses the enter key. VPG displays the text at the selected location.

10.4.6 IDENTIFY NODES/POINTS

Description: This function identifies any node and its corresponding global location in X, Y, Z coordinates. For options, see Figure 10.4.6.

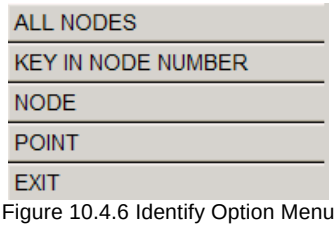


Figure 10.4.6 Identify Option Menu

Usage:

1. VPG prompts:
   > PICK NODES/POINTS
   - If the user selects ALL NODES, VPG displays the numbers of the nodes at their corresponding node locations. The user returns to the UTILITY menu.
   - If the user selects NODE, VPG displays the number of the node that is closest in proximity to the cursor. VPG issues the following message on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z =z.zzz
   - If the user selects POINT, VPG displays the number of the point that is closest in proximity to the cursor. VPG issues the following message on the command line:
     > POINT xxxx X = x.xxx Y = y.yyy Z =z.zzz
   - DONE or EXIT returns the user to the UTILITY MENU.

10.4.7 IDENTIFY ELEMENTS

Description: This function allows the user to identify an element and its nodes by cursor selection.

Usage:

1. VPG prompts:
   > SELECT ELEMENT BY CURSOR
   - VPG identifies the element's respective type, number, part and nodes. The selected element will be highlighted, and element and node numbers will be displayed in the dialogue window.
   - The user also has the option to key in the number of a desired element to identify its location and characteristics.
   - It will also identify:

NASTRAN ELEMENT TYPE	LS-DYNA ELEMENT TYPE
CBAR	BEAM
RBE2	RIGID EL
PLOTEL	PLOTEL
PLATE	PLATE
SOLID	SOLID
CELAS2	SPRING

CELAS1	SPOTWELD
RBAR	MASS
CTUBE	JOINT
ISO-SHL	T-SHELL
RBE3	SEATBELT

IDENTIFY ELEMENTS displays element numbers and corresponding node numbers.

10.4.8 LOAD CURVE

Description: This function defines and modifies the load curves for a nonlinear analysis. The LOAD CURVE command requires several levels of submenus.

Usage:

1. VPG displays the Load Curve option menu as below figure

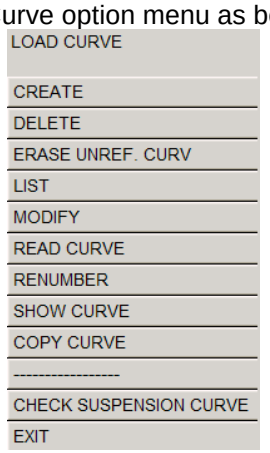


Figure 10.4.8.1 Load Curve Option Menu

2. After the user selects a command, VPG lists a set of options specific to that command.

-CREATE

Description: This function allows the user to create load curves.

Usage:

1. VPG displays the CURVE DEFINITION CARD, allowing the user to define the load curve.
   - **> ENTER LOAD CURVE NUMBER (xxxx) OR -1 TO EXIT**
   - **> ENTER CURVE NAME (UP TO 8 CHARACTERS)**
   - VPG recommends using the extension .lcr for this command.

2. VPG prompts:
   - **> ENTER DATA (TIME & VALUE) FOR POINT XXXX OR END**
   - Where XXXX is the point number and the input format is TIME, VALUE.

3. VPG displays the curve operations:

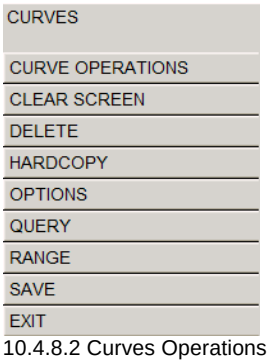


Figure 10.4.8.2 Curves Operations Menu

-DELETE

Description: This function allows the user to delete load curves.

Usage:

1. VPG displays the DEFINE CURVE CARD LIST and prompts:
   > SELECT CARD TO DELETE

2. EXIT deletes the selected load curves from the database.

-ERASE UNREF.CURVE

Description: This function will delete all functions that have not been used in model.

-LIST

Description: This function lists the available load curves in the DEFINE CURVE CARD LIST.

-MODIFY

Description: This function allows the user to quickly modify points on a defined curve.

Usage:

1. VPG lists the curves in the DEFINE CURVE CARD LIST, and prompts:
   > SELECT CURVE FROM CURVE LIST

2. After selecting the curve, the CURVE DEFINITION CARD appears, allowing the user to edit the curve data.

-READ CURVE

Description: This function imports existing curve data into the VPG database.

Usage:

1. VPG displays the VPG FILE MENU, allowing the user to enter the curve (.cur) file.

2. Once the curve file is selected and the data is read in, VPG displays the DEFINE CURVE CARD LIST. The user has the choice of selecting the curves before they are plotted.

Note: The load curve database format is shown below. ALL text must be entered with CAPITAL letters.

-LOAD CURVE FILE FORMAT

```
line 1:          $ VPG LOAD CURVE  (Header line)
line 2:          CURVE,CURVE ID,TYPE,CURVE TITLE(A5,I5,I5,1X,A8)
line 3:          X1, Y1  (2F10.4)
line 4:          X2, Y2  (2F10.4)
line 5:          X3, Y3  (2F10.4)
  etc.
line n:          CURVE,CURVEID,TYPE,CURVETITLE(Repeat a new curve)
line n+1:        X1, Y1  (2F10.4)
line n+2:        X2, Y2  (2F10.4)
line n+3:        X3, Y3  (2F10.4)
  etc.
last line:       END (A3) (Terminate Input)
```

Where:

CURVE:	Header (Required).
CURVE ID:	Load Curve Number
TYPE:	Curve Type (0- Transient Analysis; 1- Stress Initialization)
CURVE TITLE:	Curve's Title (User Defined)
X1, Y1:	Curve Point 1
X2, Y2:	Curve Point 2
END:	Header (Required)

The following is an example of the LOAD CURVE file:

```
$VPG LOAD  CURVE
CURVE   2    0 CURVE  2
     0.0000    0.0000
     1.0000    1.5000
     2.0000   -0.0000
     3.0000    0.5000
     4.0000   -1.5000
 CURVE 4    0 CURVE  4
     0.0000    2.0000
     6.0000   -3.5000
     7.0000    1.5000
     8.0000    2.0000
 END
```

-RENUMBER

Description: This function allows user to renumber load curves.

Usage:

1. VPG prompts:
   > SELECT OPTION
   ? AUTOMATIC ALL
   ONE BY ONE
   SELECT ONE
   EXIT

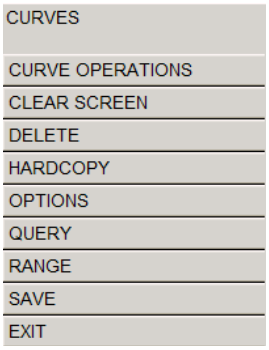
-SHOW CURVE

Description: This function displays load curves.

Usage:

1. VPG displays the DEFINE CURVE CARD LIST.

2. After selecting a curve, VPG displays the following curve operations:



These options are explained in GRAPH

-COPY CURVE

Description: This function copies load curves.

Usage:

1. VPG displays the DEFINE CURVE CARD LIST.

2. After selecting a curve or several curves, and click EXIT, copied curve(s) will be added into the list.

-CHECK SUSPENSION CURVE

Description: This function checks curves used in suspension bushing.

10.4.9 SHOW LINES

Description: This function identifies any existing line and its direction. VPG highlights the selected line and marks its points with arrowheads that are oriented in the direction of the line.

10.4.10 DATABASE STATISTICS

Description: This function displays statistics about the VPG database regarding geometry, models, materials, and interfaces, and user can export the result to an excel format file. The figures are displayed in the following window:

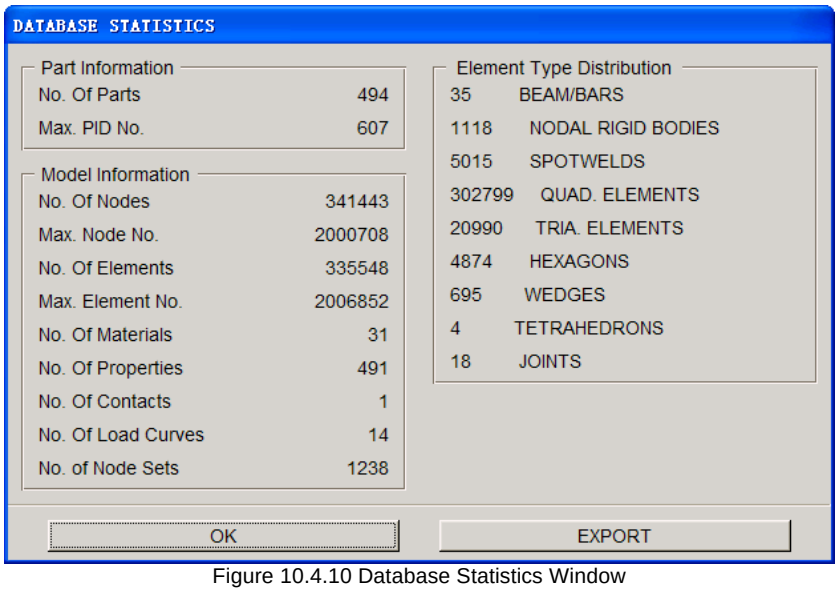


Figure 10.4.10 Database Statistics Window

10.4.11 HARDCOPY DEFAULTS

Description: This function allows the user to define the default settings for printing or saving image files. Refer the detailed descriptions in section 5.9.

10.4.12 SETUP DEFAULTS

Description: The SETUP DEFAULTS window contains various preprocessing utilities that are toggle switch activated. See the figure below.

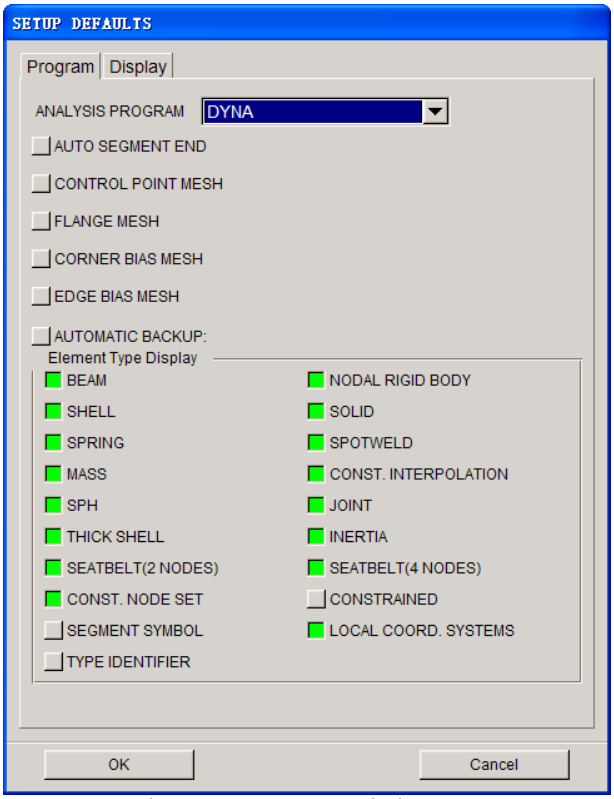
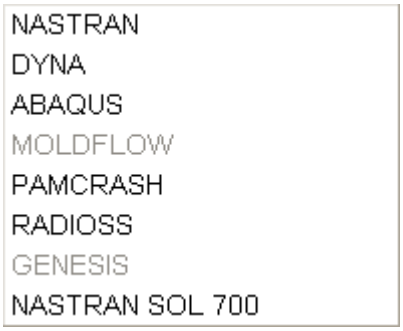


Figure 10.4.12 Setup Defaults Menu

A detailed description of each option is given in the following section

-ANALYSIS PROGRAM

Description: This option adjusts the property and material data to match the selected parameters of the solver. The default solver code for VPG is DYNA.



The MATERIAL PROPERTY and ELEMENT PROPERTY menus change according to the analysis program.

Note: VPG is unable to convert existing properties into the selected solver format.

-AUTO SEGMENT END (toggle)

Description: This option allows the user to designate the end points of line segments to serve as control points.

Note: This function is used in conjunction with CONTROL POINT MESH. VPG prompts the user to select control points on lines that are joined through the line segment option.

-CONTROL POINT MESH (toggle)

Description: This function pre-selects the points on a line for the desired node locations associated with the 2L, 3L, 4L, 6S, 8S, 9S, and 12S mesh programs.

-FLANGE MESH (toggle)

Description: This option turns on the ability of the automeshing routines to support a flange mesh option. The Flange Mesh option allows users to identify flange areas of spot welded assemblies and requires that either 2 or 3 elements be placed on the width of the flange.

-CORNER BIAS MESH (toggle)

Description: This option allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected corner of the line data that is to be modeled. This function multiplies each adjacent element that is created from the selected corner by the bias factor.

Note: A bias factor greater than 1.0 generates proportionately larger elements from the selected corner. A bias factor less than 1.0 generates proportionately smaller elements from the selected corner.

Note: This function may be used with only the 2L, 3L, 4L, 6S, 8S, 9S, and 12S functions.

-EDGE BIAS MESH (toggle)

Description: This option allows the user to enter an element bias factor (0.625 to 1.6) relative

to the selected edge of the line data that is to be modeled. This function multiplies each adjacent element that is created from the selected edge by the bias factor.

Note: A bias factor greater than 1.0 generates proportionately larger elements from the selected edge. A bias factor less than 1.0 generates proportionately smaller elements from the selected edge.

Note: This function may be used with only the 2L, 3L, 4L, 6S, 8S , 9S, and 12S functions.

-AUTOMATIC BACKUP (toggle)

Description: VPG will automatically create a backup database for the current VPG database. This parameter controls the frequency of this back-up process. The default for this parameter is OFF.

The back-up database is stored in the file named "*.*.bak", where * is the name of the current database.

In the event of a software failure or database corruption, this file may be renamed or copied and then reopened in VPG.

-ELEMENT TYPE DISPLAY

Description: The toggle switches in the ELEMENT TYPE DISPLAY determine what element types will be shown in the drawing window. The default setting for all the switches is ON.

-TYPE IDENTIFIER

Description: This toggle switch controls the display of element type identifiers. If the option is turned on, a symbol for the 1-D element types will be plotted at the center of the element to indicate the element type. The identifier symbols for the various elements are listed in the following table:

ELEMENT TYPE	ANALYSIS PROGRAM	IDENTIFIER SYMBOL
Beam	LS-DYNA & NASTRAN	**B**
CROD (Truss)	NASTRAN	**T**
CTUBE (Pipe)	NASTRAN	**P**
Rigid Element	LS-DYNA & NASTRAN	**R**
Spotweld	LS-DYNA	**W**
Discrete	LS-DYNA	**D**
Seatbelt	LS-DYNA	**S**
CELAS1 & CELAS2(Spring)	NASTRAN	**S**
Mass	LS-DYNA NASTRAN	**M**
Joint	LS-DYNA	**SPJ RVJ**

The default for this option is off.

-MENU LAYOUT

Description: This function allows the user to change Menu to VPG or FEMB style, based on the user's preference.

-MENU POSTITION

Description: This function allows the user to change Menu position to left or right hand side of the display window, based on the user's preference.

-CHANGE COLOR MAP

Description: This function allows the user to change the default colors of the COLOR BAR for plotting of part color or contour color.

-ELEMENT SIZE

Description: This function allows the user to preset a default element size before generating the 2 LINE, 3 LINE, 4 LINE, or SURFACE mesh.

-SHADING OPTIONS

Description: This function offers the following 3 options for shading a part or a model:

Options are:
GOURAUD SHADING (smooth shading)
FLAT SHADING NO EDGE
FLAT SHADING W/EDGE
FLAT SHADING W FEATURE LINE

Gouraud shading is the default setting for VPG. Flat Shading W/Edge displays the element outline in a shaded display. Flat Shading with feature line displays feature line in a shaded display, user can adjust width or color for edge and feature line.

-TRANSPARENCY FACTOR

Description: This function sets a degree of transparency for the shading option. Factors closer to 0.0 (zero) increase opacity, while factors closer to 1.0 (one) decrease opacity.

-FILL HIDE METHOD

Description: VPG provides two methods of FILLHIDE (a hidden surface plot for all of the displayed elements): STENCIL BUFFER and POLYGON OFFSET. Various graphics cards may provide an improved display appearance by using either method. It is known that on SGI IRIX workstations, specifying the STENCIL BUFFER option may result in improved graphic display performance.

-AUTOMATIC DELETE NODES WHEN DELETING ELEMENTS (toggle)

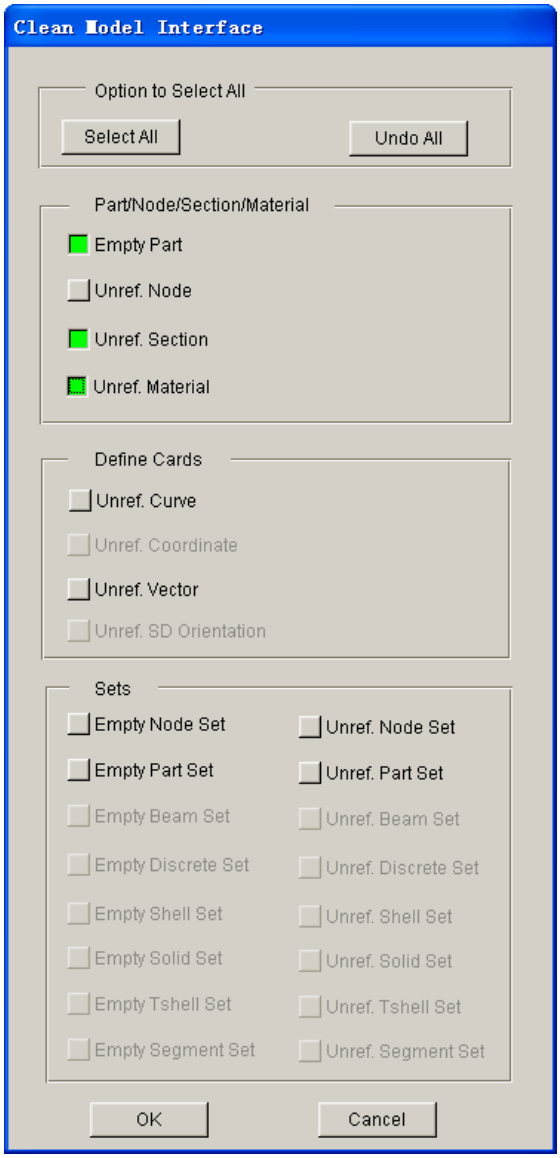
Description: This option controls when deleting elements, whether to prompt user delete nodes making up those elements. The default option is on, VPG will not prompt user whether to delete nodes, relative nodes will be deleted directly.

10.4.13 SUMMARY REPORT

Description: This function will export summary of database as report

10.4.14 CLEAN MODEL

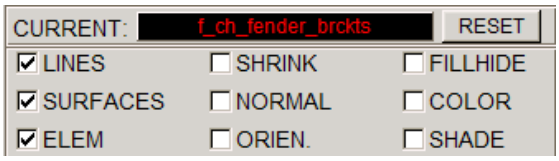
Description: This function helps to remove empty part and set, unreferenced node, section, material, define cards and sets. All these functions are integrated into a total dialog window for convenience.



If an option is shown as grey, it shows that there is no relative information in current database, for other options, user can active or inactive them as needed, user can also active all these options by method Select All. Click OK, all active options will be implemented and VPG exits this clean model function.

10.5 DISPLAY OPTIONS

Description: The functions of the DISPLAY PARAMETER OPTIONS WINDOW control various aspects of the model's appearance.



The user initiates the commands via mouse pick. The window remains active in the lower right corner of the screen. A detailed description of each function is given in the following pages.

LINES Turns plot lines on/off.

SHRINK This function reduces the size of all elements by 20 percent. SHRINK is also useful for locating any missing elements in a shell or solid structure.

FILLHIDE This function creates a hidden surface plot for all of the displayed elements, which provides the user with a realistic view of the model.

SURFACES Turns highlighted surfaces on/off.

NORMAL This function displays the orientation of an element with an arrow that is at the centroid of the element and that is perpendicular to the surface of the element. For a solid element, the arrow points toward the bottom surface of the element.

COLOR The FILLCOLOR option fills the displayed elements with a designated part color.

ELEM The ELEM option turns elements on/off.

ORIEN. This function displays element orientation. Beam element orientation is shown by drawing a vector at the first node of the beam toward the element local Y-axis. Shell orientation is shown by drawing an arrow from the first node of an element toward the second node of an element. Solid element orientation is viewed by drawing an arrow from the bottom surface toward the top surface. This display is useful for picturing the beam orientation and the normal direction of the shell elements (the normal of the shell follows the right hand rule from the first node to the second node).

Note: The clarity of the element orientation display improves when the SHRINK ELEM ON/OFF command is activated.

SHADE This command displays the elements as if they were illuminated by a light source. Elements that are not directly exposed to the light source are appropriately "shaded" to imitate actual shading.

RESET Returns the DISPLAY PARAMETER OPTIONS WINDOW to its default settings.

CURRENT PART

The current part name window not only lists the current part, but it also serves as a shortcut to execute the current part command. Clicking on this window will execute the **CURRENT PART** command, allowing the user to set a new current part. See section 10.3.

10.6 MACRO

MACRO is a module used for repeated operations. All of the user's operations will be recorded in the text file in *.xml format after activating this module. Then the user can repeat the recorded operations by executing macro files. The user can also edit the macro files to obtain the desired results. This module is very useful to reduce repeated operations and perform multiple operating system platforms.

10.6.1 START MACRO

Description: this function is used to start macro files and enter the names of the macro files. The following operations will be recorded in this macro file.

10.6.2 END MACRO

Description: this function is used to end the macro file recording and write the end mark in the macro file.

10.6.3 RUN MACRO

Description: this function is used to execute the macro files. The program returns to the initial state of the original model and executes the macro files. The recording operations in the macro file will be performed one by one.

APPENDIX A

HARDWARE AND SOFTWARE REQUIREMENTS

UNIX

PLATFORM	OS VERSION	GRAPHICS CARD	DISK SPACE (MB)
HP	HP-UX 11+	Minimum CRX 8 Plane	200
IBM	AIX 4.2+	Minimum 24 Plane Graphics	200
SGI	IRIX 6.5+	All Graphics Boards Supported	200
SUN	SunOS 5.8+	Minimum Creator 3D	200

LINUX

The RedHat operating system version 7.3 and above is supported. eta/VPG must run under KDE environment. The NVIDIA graphic cards are recommended.

PC/WINDOWS

eta/VPG in PC platform is compatible with PC-LS-DYNA 970. It will run in Windows 98, 2000, and XP environments. It is not recommended for usage with earlier versions of Windows. The following are minimum requirements for proper operation of eta/VPG in a Windows environment:

Minimum Graphics Requirement: XGA(1024 x 768)

Graphics Card: OpenGL based (Nvidia chip set recommended)

Minimum Memory Requirement:

Small models (under 50,000 elements):	384 megabytes RAM
Medium models (50,000 - 500,000 elements):	512 megabytes RAM
Large models (500,000 - 1 million elements):	768 megabytes RAM
Super large models (1 million to 4 millions):	1024+ megabytes RAM

Minimum Load space requirement: 256 megabytes
Recommended processor: Pentium 4

APPENDIX B

SUPPORTED IGES ENTITY TYPES

NAME	TYPE
Null Entity	0
Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

APPENDIX C
VPG version 3.2 RADIOSS / LS-DYNA / NASTRAN Conversion

August, 2007

OVERVIEW

The RADIOSS interface was originally released in VPG 2.0 as a complete RADIOSS interface, supporting fixed format input files from RADIOSS V2.1, V2.2, V2.3, V3.1, V4.1 and V4.4, and has the ability to write RADIOSS V3.1, V4.1 and V4.4. VPG not only reads RADIOSS files but also allows translation of RADIOSS files to NASTRAN or LS-DYNA960 format. The capability of this interface has been improved in VPG 3.2, to allow the import/export of RADIOSS and translation into LS-DYNA version 970.

When reading in a RADIOSS file VPG, adopts a RADIOSS environment with RADIOSS terminology. VPG allows model manipulation of RADIOSS materials, element properties, boundary conditions, local coordinates, loads, initial velocities, contact and rigid walls.

To translate a RADIOSS file to another analysis format such as LS-DYNA or NASTRAN, the user changes the analysis program in the Setup Menu. This will translate all RADIOSS materials, element properties, boundary conditions, etc. to the new analysis environment. VPG will prompt the user for certain user-defined information in the translation.

The translation of the code requires that certain assumptions be made in the equivalencies of elements properties, contacts, etc. Where no equivalency could be found, due to the fact that one code supports or defines an entity that another code does not, the closest substitution is made. In these cases, the user should check the database translation to be sure that the entity is defined adequately in the translated model (a complete list of all translations follows). When converting RADIOSS to LS-DYNA or vice versa, the user should always check the definition of spring elements, loading conditions, air bags, seat belt parts, dummies and barriers. These entities are either unsupported in the translation or the adequacy of the translation is questionable.

A complete list of all translations follows.

RADIOSS INTERFACE

1. The new RADIOSS interface template supports all RADIOSS cards in version 4.4, 4.1, 3.1 and 2.3.
2. VPG 3.2 allows model manipulation of RADIOSS materials, element properties, boundary conditions, local coordinates, loads, initial velocities, contact and rigid walls.
3. VPG 3.2 also supports the following functions: accelerometer, sensor, fixed velocity, section and monitored volume.
4. Rigid body and cylindrical joint elements were enhanced to use node set.
5. Multiple RADIOSS input files can be read at one time and directly read/write RADIOSS input files in DYNA970/NASTRAN database.

KEY SUMMARIES FOR CONVERSION RADIOSS TO LS-DYNA970

ELEMENT

1. Primary Nodal Rigid Body should be re-calculated by VPG 3.2.
2. Beam, Truss and Spring elements in Radioss converts to Beam element in LS-DYNA V970.

3. Joint element in Radioss convert to CONSTRAINED_NODAL_BODY and one CONSTRAINED_JOINT_CYLINDRICAL when number of node is equal 4. Otherwise VPG converts one CONSTRAINED_NODAL_RIGID_BODY.
4. Rigid Body element without mass and inertia in Radioss convert to CONSTRAINED_NODAL_RIGID_BODY, but when the Rigid Body element with mass and inertia in Radioss converts to CONSTRAINED_NODAL_RIGID_BODY_INERTIA. The Radioss primary node number(RBID) be used as the parameter NODEID on card 2 of CONSTRAINED_NODAL_RIGID_BODY_INERTIA.

ELEMENT PROPERTY

1. Delete VOID, RIVET property in Radioss.
2. BEAM converts SECTION_BEAM(RESULTANT), TRUSS converts SECTION_BEAM(TRUSS) and SPRING converts SECTION_BEAM(DISCRETE)
3.SPRING elements(TYPE 4, 8, 12 and 13) in Radioss convert into DYNA V970 material Type196(*MAT_GENERAL_SPRING_DISCRETE_BEAM). This Model permits elastic and elastoplastic spring with damping to be represented with a discrete beam element type 6. For elastic behavior, a load curve defines force or moment versus displacement or rotation. For inelastic behaivor, a load curve yield force or moment versus plastic deflection or rotation, which can vary in tension and compression.

MATERIAL PROPERTY

1. Radioss material type 2-ELASTIC_PLASTIC converts to MAT_SIMPLIFIED_JOHNSON_COOK.
2. Radioss material type 21-ELASTIC_PLASTIC_DP converts to DYNA V970 material type 57(MAT_LOW_DENSITY_FOAM).
3. Radioss material type 28-ORTHOTROPIC converts to DYNA V970 material type26 MAT_HONEYCOMB in relative SECTION_SOLID and creates new HOURGLASS card in relative PART's HGID field in DYNA970 input file. The Radioss yield functions for the HONEYCOMB material which as the abscissa values must be converted into DYNA V970 curves with abscissa values.
4. AIRBAG(MONITORED VOLUMESS) : MONITORED VOLUMES in Radioss converts SIRBAG_SIMPLE_AIRBAG_MODEL and replace the original airbag material type 27-ELASTIC_PLASTIC_BRITTLE converts to DYNA V970 material type 34 MAT_FABLIC.
5. SEATBELT with Radioss material type 1-ELASTIC converts to DYNA V970 material type 34 MAT_FABLIC with very small young's modulus.

CONTACT

1. All the Radioss contacts convert to CONTACT_AUTOMATIC_SINGLE_SURFACE contact in Dyna and create a new one more CONTACT_AUTOMATIC_SINGLE_SURFACE for full model.

VPG 3.2 RADIOSS INTERFACE AND CONVERSION TABLE

RADIOSS Interface Notes:

1. RADIOSS INTERFACE.

VPG 3.2 SUPPORTS ALL INPUT CARDS OF RADIOSS V4.1 FIXED FORMAT.

RADIOSS' Entity **Location in VPG GUI**

CONTROL VARABLES	RADIOSS MISC./CONTROL CARDS"
MATERIALS	MATERIAL PROPERTY
BOUNDARY CONDITIONS	BOUNDARY CONDITIONS/SPC
SKEW FRAMES	UTIL/COORDINATE SYSTEM
PROPERTY SETS	ELEMENT PROPERTY
FUNCTIONS	UTIL/LOAD CURVE
LOADS	BOUNDARY CONDITIONS/LOADING
INITIAL VELOCITIES	BOUNDARY CONDITIONS/INITIAL VELOCITY
ACCELEROMETERS	RADIOSS MISC./ACCELEROMETERS
SENSORS	"RADIOSS MISC./SENSORS"
GRAVITY	BOUNDARY CONDITIONS/GRAVITY"
INTERFACES	CONTACT
RIGID WALLS	CONTACT
RIGID BODIES	ELEMENT OPTIONS
ADDED MASSES	ELEMENT OPTIONS
IMPOSED VELOCITIES	BOUNDARY CONDITIONS/IMPOSED VELOCITY
RIVETS SPOTWELDS	ELEMENT OPTIONS
SECTIONS	RADIOSS MISC./SECTION CARDS
CYLINDRICAL JOINTS	ELEMENT OPTIONS
MONITORED VOLUMES	RADIOSS MISC./MONITORED VOLUMES.

2. CONVERTING FROM RADIOSS TO DYNA

2.1 ELEMENT:

RADIOSS (V4.1) ===>	DYNA (V970)
BEAM	BEAM
TRUSS	BEAM
SOLID	SOLID
SHELL	SHELL
SPRING	BEAM
MASS	MASS
JOINT	If number of node is equal 4, we will convert it into 2 CONSTRAINED_NODAL_ RIGID_BODY and 1 CONSTRAINED_JOINT_ CYLINDRICAL. Otherwise we will convert it into 1 CONSTRAINED_NODAL_RIGID_BODY.
RIGID BODY	CONSTRAINED_NODAL_RIGID_BODY (Doesn't have mass and inertia.) CONSTRAINED_NODAL_RIGID_BODY_INERTIA (Including mass and inertia. The RADIOSS primary node number (RBID) be used as the parameter NODEID on card 2 of *CONSTRAINED_NODAL_RIGID_BODY_INERTIA.)
SPOTWELD	CONSTRAINED_SPOTWELD

2.2 ELEMENT PROPERTY:

Delete VOID, RIVET property. Convert spring stiffness into DYNA material.

RADIOSS (V4.1) ===>	DYNA (V970)
1 -SHELL	SECTION_SHELL
2 -TRUSS	SECTION_BEAM (truss)
3 -BEAM	SECTION_BEAM (resultant)
4 -SPRING	SECTION_BEAM (discrete)
6 -ORTHOTROPIC SOLID	SECTION_SOLID
8 -GENERAL SPRING	SECTION_BEAM (discrete)
9 -ORTHOTROPIC SHELL	SECTION_SHELL
10-COMP. SHELL	SECTION_SHELL
11-COMP. SHELL	SECTION_SHELL
12-3-NODES SPRING	SECTION_BEAM (discrete)

13-BEAM TYPE SPRING SECTION_BEAM (discrete)
14-GENERAL SOLID SECTION_SOLID

CONVERTS RADIOSS SPRING PROPERTY(TYPE 4,8,12,13) INTO DYNA's MATERIAL 196.

IF H=0, N1=0: LINEAR ELASTIC SPRING
K(DYNA) = K(RADIOSS)
D(DYNA) = C(RADIOSS)
IF H=0, N1>0: NONLINEAR ELASTIC SPRING
K(DYNA) = A(RADIOSS) (IF A(RADIOSS)=0.0, THEN K(DYNA)=1.0)
C2(DYNA) = B(RADIOSS)/A(RADIOSS)
FLCID=GLCID = N1(RADIOSS)
HLCID = N2(RADIOSS)
DLE(DYNA) = D(RADIOSS)
IF H=1, N1>0:
K(DYNA) = K(RADIOSS)
A(RADIOSS) = 1.0 IS ASSUMED
C2(DYNA) = B(RADIOSS)
FLCID=GLCID = N1(RADIOSS)
HLCID = N2(RADIOSS)
DLE(DYNA) = D(RADIOSS)

CONVERT RADIOSS SPRING INTO DYNA BEAM.

IN ALL CASES, IF THE LENGTH OF SPRING IS FINITE, SET SCOOR IN
*SECTION_BEAM TO 2. IF THE LENGTH IS VERY SMALL(<=0.000001) OR ZERO,
SET SCOOR TO 0.

2.3 MATERIAL PROPERTY:

RADIOSS(V4.1) ===> DYNA(V970)

0 -VOID 1 -MAT_ELASTIC
1 -ELASTIC 1 -MAT_ELASTIC
2 -ELASTIC_PLASTIC 98-MAT_SIMPLIFIED_JOHNSON_COOK
3 -ELASTIC_PLASTIC_HYDRODYNAMIC 10-MAT_ELASTIC_PLASTIC_HYDRO
4 -JOHNSON_COOK 15-MAT_JOHNSON_COOK
6 -HYDRODYNAMIC_VISCOUS 9 -MAT_NULL
10-ELASTIC_PLASTIC_DP 63-MAT_CRUSHABLE_FOAM
14-ELASTIC_PLASTIC_ORTHOTROPIC 2 -MAT_ORTHOTROPIC_ELASTIC
19-ELASTIC_ORTHOTROPIC 130 -MAT_SPECIAL_ORTHOTROPIC
21-ELASTIC_PLASTIC_DP 57-MAT_LOW_DENSITY_FOAM
There is no direct conversion of RADIOSS mat 21 to *MAT_LOW_DENSITY_FOAM.
An example input for a mat57 seat foam is attached.

```
$ UNITS:    kg, mm, ms, GPa
*MAT_LOW_DENSITY_FOAM
$    seat foam
       571258.8200E-08       0.0045         2222          0.4          0.0          0.0          .05
          0.0          0.0          0.0          0.0          0.0          0.0
*DEFINE_CURVE
         2222            0          1.0         0.01          0.0          0.0            0
    0.0                    0.0
    0.0625                 0.119800001
    0.122500002            0.383390009
    0.177499995            0.377400011
    0.234999999            0.413399994
    0.280699998            0.444999993
    0.344999999            0.466100007
    0.397000015            0.497999996
    0.444900006            0.53670001
    0.49000001             0.561600029
    0.528999984            0.601000011
    0.564999998            0.653999984
    0.597000003            0.721000016
    0.622500002            0.818000019
    0.644900024            0.897000015
    0.660000026            0.949999988
    0.666999996            0.989000022
```

0.670000017	0.99000001
0.671999991	1.0
0.699000001	1.20000005
0.75	1.51999998
0.800000012	2.19000006
0.808000028	2.30299997

22-ELASTIC_PLASTIC	81-MAT_PLASTICITY_WITH_DAMAGE
23-ELASTIC_PLASTIC	81-MAT_PLASTICITY_WITH_DAMAGE
24-ELASTIC_PLASTIC_BRITTLE	16-MAT_PSEUDO_TENSOR
25-ELASTIC_PLASTIC_ORTHOTROPIC	2 -MAT_ORTHOTROPIC_ELASTIC
27-ELASTIC_PLASTIC_BRITTLE	81-MAT_PLASTICITY_WITH_DAMAGE
28-ORTHOTROPIC	26-MAT_HONEYCOMB

We set DYNA's value PR=0.0, SIGY=1.0, VF=0.2, MU=0.2, and ELFORM=1 in relative *SECTION_SOLID card. We create a *HOURGLASS card with IHQ=4, and put it into relative *PART's HGID field.

In RADIOSS, if the relative element property ID is 0 or the element property type is not TYPE 6, we will set the material directions 1,2,3 align with the global axes X,Y,Z, AOPT=2, (A1,A2,A3)=(1,0,0), (D1,D2,D3)=(0,1,0). Otherwise we will set AOPT=0, and map the RADIOSS 1,2,3 axes to the LS-DYNA b,c,a axes. The RADIOSS yield functions for the honeycomb material which have mu as the abscissa values must be converted into LS-DYNA curves with mu/(1+mu) as the abscissa values. In the discussion above, it is assumed that the angle PHI in the RADIOSS orthotropic solid property set data is set to zero. If PHI is nonzero, further thought must be given to the conversion.

32-ELASTIC_PLASTIC_ORTHOTROPIC	3 -MAT_PLASTIC_KINEMATIC
33-VISCOPLASTIC	53-MAT_CLOSED_CELL_FOAM
34-VISCOELASTIC	61-MAT_KELVIN-MAXWELL_VISCOELASTIC
35-VISCOELASTIC	76-MAT_GENERAL_VISCOELASTIC
36-ELASTIC_PLASTIC	24-MAT_PIECEWISE_LINEAR_ PLASTICITY
38-VISCOELASTIC	76-MAT_GENERAL_VISCOELASTIC
40-VISCOELASTIC	76-MAT_GENERAL_VISCOELASTIC
42-HYPERELASTIC	77(5)-MAT_OGDEN_RUBBER
43-ELASTIC_PLASTIC_ORTHOTROPIC	24-MAT_PIECEWISE_LINEAR_ PLASTICITY

2.4 BOUNDARY AND LOAD CONDITION:

RADIOSS (V4.1) ===>	DYNA (V970)
SPC	BOUNDARY_SPC_SET
CONCENTRATED LOAD	LOAD_NODE_SET
PRESSURE LOAD	LOAD_SEGMENT_SET
INITIAL VELOCITY	INITIAL_VELOCITY
IMPOSED VELOCITY	BOUNDARY_PRESCRIBED_MOTION_SET

2.5 CONTACT AND RIGIDWALL:

Delete RIGIDWALL which not include node.
Convert slave/master shell material/property set into segment set.

RADIOSS(V4.1) ===>	DYNA(V970)
2 -TIED	CONTACT_AUTOMATIC_SINGLE_SURFACE
3 -SLIDE/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE
5 -SLIDE/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE
6 -SLIDE/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE
7 -SLIDE/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE
8 -SLIDE	CONTACT_DRAWBEAD
10-TIED/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE
11-SLIDE/VOID	CONTACT_AUTOMATIC_SINGLE_SURFACE

Creates a new CONTACT_AUTOMATIC_SINGLE_SURFACE.
Set all CONTACT_AUTOMATIC_SINGLE_SURFACE's SOFT=1, IGNORE=1.

RIGIDWALL	RIGIDWALL_GEOMETRIC

2.6 AIRBAG (MONITORED VOLUMES):

MONITORED VOLUMES	AIRBAG_SIMPLE_AIRBAG_MODEL

REPLACES THE ORIGINAL MATERIAL WITH *MAT_FABRIC.

2.7 SEATBELT:

CONVERTS RADIOSS'S MATERIAL TYPE 1 WITH VERY SMALL YOUNG'S MODULUS INTO DYNA'S *MAT_FABRIC.

2.8 OTHERS:

Delete TRUSS SET, MAT SET, PROP SET.
Delete ACCELEROMETER, MONITORED VOLUME, SECTIONS,
SENSOR, CONTROL CARDS.

RADIOSS (V4.1) ===> DYNA (V970)

FUNCTION	DEFINE_CURVE
SKEW (moving)	DEFINE_COORDINATE_NODES
SKEW (fixed)	DEFINE_COORDINATE_VECTOR

3. CONVERTING FROM DYNA TO RADIOSS

3.1 ELEMENT:

Delete INERTIA elements.

DYNA (V970) ===> RADIOSS (V4.1)

BEAM	BEAM
SOLID	SOLID
SHELL	SHELL
T-SHELL	SOLID
DISCRETE	SPRING
(VID is converted into the third node)	
MASS	MASS
JOINT	JOINT
CONSTRAINED_NODAL_RIGID_BODY	RIGID BODY
CONSTRAINED_NODE_SET	RIGID BODY
CONSTRAINED_SPOTWELD	SPOTWELD
SEATBELT	BEAM

3.2 ELEMENT PROPERTY:

Delete SPH section.
Convert spring stiffness into RADIOSS element property.

DYNA (V970) ===> RADIOSS (V4.1)

SECTION_BEAM (integrated)	3 -BEAM
SECTION_BEAM (resultant)	3 -BEAM
SECTION_BEAM (discrete)	3 -BEAM
SECTION_BEAM (truss)	3 -BEAM
SECTION_BEAM (spotweld)	3 -BEAM
SECTION_DISCRETE	4 -SPRING
SECTION_SHELL	1 -SHELL
SECTION_SHELL_ALE	1 -SHELL
SECTION_SOLID	14-GENERAL SOLID
SECTION_SOLID_ALE	14-GENERAL SOLID
SECTION_TSHELL	14-GENERAL SOLID
SECTION_SEATBELT	3 -BEAM

3.3 MATERIAL PROPERTY:

DYNA (V970) ===> RADIOSS (V4.1)

1 -MAT_ELASTIC	1 -ELASTIC
1 -MAT_ELASTIC_FLUID	1 -ELASTIC
2 -MAT_ORTHOTROPIC_ELASTIC	14-ELASTIC_PLASTIC_ORTHOTROPIC

2 -MAT_ANISOTROPIC_ELASTIC	1 -ELASTIC
3 -MAT_PLASTIC_KINEMATIC	2 -ELASTIC_PLASTIC
4 -MAT_ELASTIC_PLASTIC_THERMAL	2 -ELASTIC_PLASTIC
5 -MAT_SOIL_AND_FOAM	33-VISCOPLASTIC
6 -MAT_VISCOELASTIC	34-VISCOELASTIC
7 -MAT_BLATZ-KO_RUBBER	1 -ELASTIC
8 -MAT_HIGH_EXPLOSIVE_BURN	1 -ELASTIC
9 -MAT_NULL	6 -HYDRODYNAMIC_VISCOUS
10-MAT_ELASTIC_PLASTIC_HYDRO	3 -ELASTIC_PLASTIC_HYDRODYNAMIC
10(2) - MAT_ELASTIC_PLASTIC_HYDRO_SPALL	3 -ELASTIC_PLASTIC_HYDRODYNAMIC
11-MAT_STEINBERG	1 -ELASTIC
11(2) - MAT_STEINBERG_LUND	1 -ELASTIC
12-MAT_ISOTROPIC_ELASTIC_PLASTIC	2 -ELASTIC_PLASTIC
13-MAT_ISOTROPIC_ELASTIC_FAILURE	2 -ELASTIC_PLASTIC
14-MAT_SOIL_AND_FOAM_FAILURE	33-VISCOPLASTIC
15-MAT_JOHNSON_COOK	4 -JOHNSON_COOK
16-MAT_PSEUDO_TENSOR	24-ELASTIC_PLASTIC_BRITTLE
17-MAT_ORIENTED_CRACK	2 -ELASTIC_PLASTIC
18-MAT_POWER_LAW_PLASTICITY	2 -ELASTIC_PLASTIC
19-MAT_STRAIN_RATE_DEPENDENT_PLASTICITY	2 -ELASTIC_PLASTIC
20-MAT_RIGID	1 -ELASTIC
21-MAT_ORTHOTROPIC_THERMAL	14-ELASTIC_PLASTIC_ORTHOTROPIC
22-MAT_COMPOSITE_DAMAGE	14-ELASTIC_PLASTIC_ORTHOTROPIC
23-MAT_TEMPERATURE_DEPENDENT _ORTHOTROPIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
24-MAT_PIECEWISE_LINEAR_PLASTICITY	36-ELASTIC_PLASTIC
25-MAT_GEOLOGICAL_CAP_MODEL	24-ELASTIC_PLASTIC_BRITTLE
26-MAT_HONEYCOMB	2 -ELASTIC_PLASTIC
27-MAT_MOONEY-RIVLIN_RUBBER	1 -ELASTIC
28-MAT_RESULTANT_PLASTICITY	2 -ELASTIC_PLASTIC
29-MAT_FORCE_LIMITED	2 -ELASTIC_PLASTIC
30-MAT_CLOSED_FORM_SHELL _PLASTICITY	2 -ELASTIC_PLASTIC
31-MAT_FRAZER_NASH_RUBBER_MODEL	1 -ELASTIC
32-MAT_LAMINATED_GLASS	2 -ELASTIC_PLASTIC
33-MAT_BARLAT_ANISOTROPIC_PLASTICITY	2 -ELASTIC_PLASTIC
33(2) - MAT_BARLAT_YLD96	2 -ELASTIC_PLASTIC
34-MAT_FABRIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
35-MAT_PLASTIC_GREEN-NAGHDI_RATE	2 -ELASTIC_PLASTIC
36-MAT_3-PARAMETER_BARLAT	2 -ELASTIC_PLASTIC
37-MAT_TRANSVERSELY_ANISOTROPIC _ELASTIC_PLASTIC	2 -ELASTIC_PLASTIC
38-MAT_BLATZ-KO_FOAM	33-VISCOPLASTIC
39-MAT_FLD_TRANSVERSELY_ANISOTROPIC	2 -ELASTIC_PLASTIC
40-MAT_NONLINEAR_ORTHOTROPIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
41-MAT_USER_DEFINED_MATERIAL_MODELS	1 -ELASTIC
51-MAT_BAMMAN	2 -ELASTIC_PLASTIC
52-MAT_BAMMAN_DAMAGE	2 -ELASTIC_PLASTIC
53-MAT_CLOSED_CELL_FOAM	33-VISCOPLASTIC
54-MAT_ENHANCED_COMPOSITE_DAMAGE	14-ELASTIC_PLASTIC_ORTHOTROPIC
57-MAT_LOW_DENSITY_FOAM	33-VISCOPLASTIC
58-MAT_LAMINATED_COMPOSITE_FABRIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
59-MAT_COMPOSITE_FAILURE_SHELL_MODEL	14-ELASTIC_PLASTIC_ORTHOTROPIC
59-MAT_COMPOSITE_FAILURE_SOLID_MODEL	14-ELASTIC_PLASTIC_ORTHOTROPIC
60-MAT_ELASTIC_WITH_VISCOSITY	1 -ELASTIC
61-MAT_KELVIN-MAXWELL_VISCOELASTIC	34-VISCOELASTIC
62-MAT_VISCOUS_FOAM	33-VISCOPLASTIC
63-MAT_CRUSHABLE_FOAM	10-ELASTIC_PLASTIC_DP
64-MAT_RATE_SENSITIVE_POWERLAW	2 -ELASTIC_PLASTIC_PLASTICITY
65-MAT_MODIFIED_ZERILLI _ARMSTRONG	2 -ELASTIC_PLASTIC
66-MAT_LINEAR_ELASTIC_DISCRETE_BEAM	1 -ELASTIC
67-MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM	1 -ELASTIC
68-MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM	2 -ELASTIC_PLASTIC
69-MAT_SID_DAMPER_DISCRETE_BEAM	1 -ELASTIC
70-MAT_HYDRAULIC_GAS_DAMPER	1 -ELASTIC_DISCRETE_BEAM
71-MAT_CABLE_DISCRETE_BEAM	1 -ELASTIC
72-MAT_CONCRETE_DAMAGE	24-ELASTIC_PLASTIC_BRITTLE
73-MAT_LOW_DENSITY_VISCOUS_FOAM	33-VISCOPLASTIC
74-MAT_ELASTIC_SPRING_DISCRETE_BEAM	1 -ELASTIC
75-MAT_BILKHU/DUBOIS_FOAM	33-VISCOPLASTIC
76-MAT_GENERAL_VISCOELASTIC	35-VISCOELASTIC

77-MAT_HYPERELASTIC_RUBBER	42-HYPERELASTIC
77-MAT_OGDEN_RUBBER	42-HYPERELASTIC
78-MAT_SOIL_CONCRETE	10-ELASTIC_PLASTIC_DP
79-MAT_HYSTERETIC_SOIL	24-ELASTIC_PLASTIC_BRITTLE
80-MAT_RAMBERG_OSGOOD	1 -ELASTIC
81-MAT_PLASTICITY_WITH_DAMAGE	22-ELASTIC_PLASTIC
83-MAT_FU_CHANG_FOAM	33-VISCOPLASTIC
86-MAT_ORTHOTROPIC_VISCOELASTIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
87-MAT_CELLULAR_RUBBER	1 -ELASTIC
88-MAT_MTS	1 -ELASTIC
89-MAT_PLASTICITY_POLYMER	2 -ELASTIC_PLASTIC
90-MAT_ACOUSTIC	1 -ELASTIC
94-MAT_INELASTIC_SPRING_DISCRETE_BEAM	1 -ELASTIC
96-MAT_BRITTLE_DAMAGE	24-ELASTIC_PLASTIC_BRITTLE
98-MAT_SIMPLIFIED_JOHNSON_COOK	2 -ELASTIC_PLASTIC
100-MAT_SPOTWELD	1 -ELASTIC
103-MAT_ANISOTROPIC_VISCOPLASTIC	33-VISCOPLASTIC
104-MAT_DAMAGE_1	22-ELASTIC_PLASTIC
105-MAT_DAMAGE_2	22-ELASTIC_PLASTIC
116-MAT_COMPOSITE_LAYUP	14-ELASTIC_PLASTIC_ORTHOTROPIC
117-MAT_COMPOSITE_MATRIX	1 -ELASTIC
118-MAT_COMPOSITE_DIRECT	1 -ELASTIC
123-MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY	36-ELASTIC_PLASTIC
124-MAT_PLASTICITY_COMPRESSION_TENSION	2 -ELASTIC_PLASTIC
126-MAT_MODIFIED_HONEYCOMB	2 -ELASTIC_PLASTIC
127-MAT_ARRUDA_BOYCE_RUBBER	42-HYPERELASTIC
128-MAT_HEART_TISSUE	1 -ELASTIC
129-MAT_LUNG_TISSUE	1 -ELASTIC
130-MAT_SPECIAL_ORTHOTROPIC	19-ELASTIC_ORTHOTROPIC
OTHER MATERIAL	1 -ELASTIC

3.4 BOUNDARY AND LOAD CONDITION:

DYNA (V970) ===>	RADIOSS (V4.1)
BOUNDARY_SPC_NODE	SPC
BOUNDARY_SPC_SET	SPC
INITIAL_VELOCITY	INITIAL VELOCITY
INITIAL_VELOCITY_NODE	INITIAL VELOCITY
INITIAL_VELOCITY_GENERATION	INITIAL VELOCITY
LOAD_NODE_POINT (Deleted, when DOF is equal to 4 or 8)	CONCENTRATED LOAD
LOAD_NODE_SET (Deleted, when DOF is equal to 4 or 8)	CONCENTRATED LOAD
LOAD_SEGMENT	PRESSURE LOAD
LOAD_SEGMENT_SET	PRESSURE LOAD
LOAD_SHELL_ELEMENT	PRESSURE LOAD
LOAD_SHELL_SET	PRESSURE LOAD
BOUNDARY_PRESCRIBED_MOTION	IMPOSED VELOCITY_NODE (velocity)
BOUNDARY_PRESCRIBED_MOTION	IMPOSED VELOCITY_RIGID (velocity)
BOUNDARY_PRESCRIBED_MOTION	IMPOSED VELOCITY_RIGID_LOCAL (velocity)
BOUNDARY_PRESCRIBED_MOTION	IMPOSED VELOCITY_SET (velocity)

3.5 CONTACT AND RIGIDWALL:

Delete CONTACT_ENTITY, CONTACT_INTERIOR, CONTACT_1D,
CONTACT_2D_AUTOMATIC_SINGLE_SURFACE,
CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE,
CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE,
CONTACT_2D_AUTOMATIC_SURFACE_IN_CONTINUUM,
CONTACT_2D_PENALTY,
CONTACT_2D_PENALTY_FRICTION,
CONTACT_2D_SLIDING_ONLY,
CONTACT_2D_SLIDING_VOIDS,
CONTACT_2D_TIED_SLIDING,
CONTACT_GEBOD.
Convert slave type 2,3,5,6 into segment set.
Convert master type 2,3 into segment set.

DYNA (V970) ===>	RADIOSS (V4.1)
CONTACT_AIRBAG_SINGLE_SURFACE	7 -SLIDE/VOID
CONTACT_AUTOMATIC_GENERAL	7 -SLIDE/VOID
CONTACT_AUTOMATIC_NODES_TO	7 -SLIDE/VOID_SURFACE
CONTACT_AUTOMATIC_ONE_WAY	7 -SLIDE/VOID_SURFACE_TO_SURFACE
CONTACT_AUTOMATIC_SINGLE	7 -SLIDE/VOID_SURFACE
CONTACT_AUTOMATIC_SURFACE	7 -SLIDE/VOID_TO_SURFACE
CONTACT_CONSTRAINT_NODES	7 -SLIDE/VOID_TO_SURFACE
CONTACT_CONSTRAINT_SURFACE	7 -SLIDE/VOID_TO_SURFACE
CONTACT_DRAWBEAD	8 -SLIDE
CONTACT_ERODING_NODES_TO_SURFACE	7 -SLIDE/VOID
CONTACT_ERODING_SINGLE_SURFACE	7 -SLIDE/VOID
CONTACT_ERODING_SURFACE_TO	7 -SLIDE/VOID_SURFACE
CONTACT_FORCE_TRANSDUCER	7 -SLIDE/VOID_PENALTY
CONTACT_FORCE_TRANSDUCER	7 -SLIDE/VOID_CONSTRAINT
CONTACT_FORMING_NODES_TO_SURFACE	7 -SLIDE/VOID
CONTACT_FORMING_ONE_WAY	7 -SLIDE/VOID_SURFACE_TO_SURFACE
CONTACT_FORMING_SURFACE_TO_SURFACE	7 -SLIDE/VOID
CONTACT_NODES_TO_SURFACE	7 -SLIDE/VOID
CONTACT_ONE_WAY_SURFACE_TO	7 -SLIDE/VOID_SURFACE
CONTACT_RIGID_NODES_TO_RIGID_BODY	7 -SLIDE/VOID
CONTACT_RIGID_BODY_ONE_WAY_TO	6 -SLIDE/VOID_RIGID_BODY
CONTACT_RIGID_BODY_TWO_WAY_TO	6 -SLIDE/VOID_RIGID_BODY
CONTACT_SINGLE_EDGE	7 -SLIDE/VOID
CONTACT_SINGLE_SURFACE	7 -SLIDE/VOID
CONTACT_SLIDING_ONLY	7 -SLIDE/VOID
CONTACT_SLIDING_ONLY_PENALTY	7 -SLIDE/VOID
CONTACT_SURFACE_TO_SURFACE	7 -SLIDE/VOID
CONTACT_TIEBREAK_NODES_TO	7 -SLIDE/VOID_SURFACE
CONTACT_TIEBREAK_SURFACE_TO	7 -SLIDE/VOID_SURFACE
CONTACT_TIED_NODES_TO_SURFACE	7 -SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO	7 -SLIDE/VOID_SURFACE
CONTACT_TIED_SURFACE_TO_SURFACE	2 -TIED
CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE	2 -TIED
RIGIDWALL	RIGIDWALL

3.6 OTHERS:

Delete T-SHELL SET.
Delete all other cards that can't be converted.

DYNA (V970) ===>	RADIOSS (V4.1)
DEFINE_COORDINATE_NODES	SKEW_MOV
DEFINE_COORDINATE_SYSTEM	SKEW_MOV
DEFINE_COORDINATE_VECTOR	SKEW_FIX
DEFINE_CURVE	FUNCTION

4. CONVERTING FROM NASTRAN TO DYNA

4.1 ELEMENT:

General Comments: When some RBE2s share the common nodes, merge them into one RBE2. Delete RBE3 & CGAP elements.

NASTRAN ===>	DYNA (V970)
CBAR	BEAM
RBE2	CONSTRAINED_NODAL_RIGID_BODY, CONSTRAINED_NODE_SET (According to the Dof)
SHELL	SHELL
SOLID	SOLID

CELAS2	DISCRETE (Create DEFINE_SD_ORIENTATION to define the spring direction)
CELAS1	DISCRETE (Create DEFINE_SD_ORIENTATION to define the spring direction)
RBAR	CONSTRAINED_SPOTWELD
CONM2	MASS & INERTIA
CBEAM	BEAM
CROD	BEAM
CTUBE	BEAM
ISOPARAMETRIC SHELL	SHELL

4.2 ELEMENT PROPERTY:

General Comments: Delete PGAP properties. Convert spring stiffness into DYNA material.

NASTRAN	===>	DYNA (V970)
PSHELL		SECTION_SHELL
PSOLID		SECTION_SOLID
PBAR		SECTION_BEAM (resultant)
PBEAM		SECTION_BEAM (resultant)
PELAS		SECTION_DISCRETE
PROD		SECTION_BEAM (truss)
PTUBE		SECTION_BEAM (integrated)

4.3 MATERIAL PROPERTY:

NASTRAN	===>	DYNA (V970)
MAT1		1 -MAT_ELASTIC

4.4 BOUNDARY AND LOAD CONDITION:

NASTRAN	===>	DYNA(V970)
SPC		BOUNDARY_SPC_SET
NODE FORCE		LOAD_NODE_SET
SHELL PRESSURE		LOAD_SHELL_SET
INITIAL VELOCITY		INITIAL_VELOCITY

4.5 OTHERS:

General Comments: Convert NODE SET, ELEMENT SET.

NASTRAN	===>	DYNA (V970)
LOAD CURVE		DEFINE_CURVE
LOCAL COORDINATE		DEFINE_COORDINATE_SYSTEM

5. CONVERTING FROM DYNA TO NASTRAN

5.1 ELEMENT:

General Comments: Delete SEATBELT & JOINT elements.

DYNA (V970)	===>	NASTRAN
BEAM		CBAR, CBEAM, CROD,CTUBE (according to element Prop.)
CONSTRAINED_NODAL_RIGID_BODY		RBE2
T-SHELL		SOLID
SHELL		SHELL

SOLID	SOLID	
DISCRETE	CELAS1	(Create LCS for spring direction)
MASS	CONM2	
CONSTRAINED_SPOTWELD	RBAR	
INERTIA	CONM2	
CONSTRAINED_NODE_SET	RBE2	

5.2 ELEMENT PROPERTY:

General Comments: Delete SEATBELT section. Convert spring stiffness into element property.

DYNA (V970)	===>	NASTRAN
SECTION_BEAM (integrated)		PBAR, PTUBE (According to ELFORM and CST)
SECTION_BEAM (resultant)		PBAR
SECTION_BEAM (discrete)		PBAR
SECTION_BEAM (truss)		PROD
SECTION_BEAM (spotweld)		PBAR
SECTION_DISCRETE		PELAS
SECTION_SHELL		PSHELL
SECTION_SOLID		PSOLID
SECTION_SOLID_ALE		PSOLID
SECTION_TSHELL		PSOLID

5.3 MATERIAL PROPERTY:

All materials	MAT1.

5.4 BOUNDARY AND LOAD CONDITION:

DYNA (V970)	===>	NASTRAN
BOUNDARY_SPC_NODE		SPC
BOUNDARY_SPC_SET		SPC
INITIAL_VELOCITY		INITIAL VELOCITY
INITIAL_VELOCITY_NODE		INITIAL VELOCITY
INITIAL_VELOCITY_GENERATION		INITIAL VELOCITY
LOAD_NODE_POINT		NODAL FORCE
LOAD_NODE_SET		NODAL FORCE
LOAD_SHELL_ELEMENT		SHELL PRESSURE
LOAD_SHELL_SET		SHELL PRESSURE

5.5 OTHERS:

General Comments: Convert NODE SET, ELEMENT SET. Delete all other cards that can't be converted.

DYNA (V970)	===>	NASTRAN
DEFINE_COORDINATE_NODES		LOCAL COORDINATE
DEFINE_COORDINATE_SYSTEM		LOCAL COORDINATE
DEFINE_COORDINATE_VECTOR		LOCAL COORDINATE
DEFINE_CURVE		LOAD CURVE

6. CONVERTING FROM RADIOSS TO NASTRAN

6.1 ELEMENT:

RADIOSS (V4.1)	===>	NASTRAN
BEAM		CBAR

TRUSS	CROD
SOLID	SOLID
SHELL	SHELL
SPRING	ELAS1(3rd node is converted to a LCS in NASTRAN)
MASS	CONM2
JOINT	RBE2
RIGID BODY	RBE2
SPOTWELD	RBAR

6.2 ELEMENT PROPERTY:

General Comments: Delete VOID, RIVET properties.

RADIOSS (V4.1)	===>	NASTRAN
1 -SHELL		PSHELL
2 -TRUSS		PROD
3 -BEAM		PBAR
4 -SPRING		PELAS
6 -ORTHOTROPIC SOLID		PSOLID
8 -GENERAL SPRING		PELAS
9 -ORTHOTROPIC SHELL		PSHELL
10-COMP. SHELL		PSHELL
11-COMP. SHELL		PSHELL
12-3-NODES SPRING		PELAS
13-BEAM TYPE SPRING		PELAS
14-GENERAL SOLID		PSOLID

6.3 MATERIAL PROPERTY:

All materials	MAT1

6.4 BOUNDARY AND LOAD CONDITION:

General Comments: Delete PRESSURE LOAD.

RADIOSS (V4.1)	===>	NASTRAN
SPC		SPC
CONCENTRATED LOAD		NODE FORCE
INITIAL VELOCITY		INITIAL VELOCITY

6.5 OTHERS:

General Comments: Delete NODE SET, SHELL SET, BEAM SET, SOLID SET, SPRING SET, TRUSS SET. Delete all others cards that can't be converted.

RADIOSS (V4.1)	===>	NASTRAN
FUNCTION		LOAD CURVE
SKEW (moving)		LOCAL COORDINATE
SKEW (fixed)		LOCAL COORDINATE

7. CONVERTING FROM NASTRAN TO RADIOSS

7.1 ELEMENT:

General Comments: Delete RBE3 & CGAP elements.

NASTRAN	===>	RADIOSS (V4.1)
CBAR		BEAM
RBE2		RIGID BODY
SHELL		SHELL
SOLID		SOLID
CELAS2		SPRING(3rd node is created if necessary to define the direction)
CELAS1		SPRING(3rd node is created if necessary to define the direction)
RBAR		SPOTWELD
CONM2		MASS
CBEAM		BEAM
CROD		TRUSS
CTUBE		BEAM
ISOPARAMETRIC SHELL		SHELL

7.2 ELEMENT PROPERTY:

General Comments: Delete PGAP properties.

NASTRAN	===>	RADIOSS (V4.1)
PSHELL		1 -SHELL
PSOLID		14-GENERAL_SOLID
PBAR		3 -BEAM
PBEAM		3 -BEAM
PELAS		4 -SPRING
PROD		2 -TRUSS
PTUBE		3 -BEAM

7.3 MATERIAL PROPERTY:

NASTRAN	===>	RADIOSS (V4.1)
MAT1		1 -ELASTIC

7.4 BOUNDARY AND LOAD CONDITION:

NASTRAN	===>	RADIOSS (V4.1)
SPC		SPC
NODE FORCE		CONCENTRATED LOAD
SHELL PRESSURE		PRESSURE LOAD
INITIAL VELOCITY		INITIAL_VELOCITY

7.5 OTHERS:

General Comments: Delete NODE SET, ELEMENT SET.

NASTRAN	===>	RADIOSS (V4.1)
LOAD CURVE		FUNCTION
LOCAL COORDINATE SYSTEM		SKEW_MOV

RADIOSS 4.4 Conversion to LS-DYNA & NASTRAN

8. CONVERTING FROM RADIOSS TO DYNA.

8.1 ELEMENT:

RADIOSS(V4.4)	===>	DYNA(V970)
BEAM		BEAM
JOINT		If RADIOSS's JOINT has 4 nodes, VPG will convert it into two *CONSTRAINED_NODAL_RIGID_BODY and one *CONSTRAINED_JOINT_CYLINDRICAL. Otherwise, VPG will convert it into *CONSTRAINED_NODAL_RIGID_BODY.
MASS		MASS
QUAD		SHELL
RBODY		If RADIOSS RIGID BODY does not have mass and inertia, VPG will convert it into *CONSTRAINED_NODAL_RIGID_BODY. Otherwise, VPG will convert it into *CONSTRAINED_NODAL_RIGID_BODY_INERTIA. RADIOSS's primary node(RBID) will be used as DYNA's "NODEID" in second card of *CONSTRAINED_NODAL_RIGID_BODY_INERTIA.
RIVET		*CONSTRAINED_SPOTWELD If user need MAT100 spotweld instead Of *CONSTRAINED_SPOTWELD, user can use function "SPOTWELD TO MAT100 SP." under "ELEMENT OPTION/MODIFY/TYPE" to convert it.
RLINK		*CONSTRAINED_NODAL_RIGID_BODY
SHELL		SHELL
SHEL16		SOLID
SOLID		SOLID
SPRING		If this spring has element property type 4, 8, 12, 13, VPG will convert it into BEAM. Otherwise, VPG convert it into spring.
TRUSS		BEAM
XELEM		*CONSTRAINED_NODAL_RIGID_BODY

8.2 ELEMENT PROPERTY:

Delete VOID, RIVET, KJOINT, NSTRAND property.
Convert spring stiffness into DYNA material.

RADIOSS (V4.4)	===>	DYNA (V970)
1 -SHELL		SECTION_SHELL
2 -TRUSS		SECTION_BEAM(truss)
3 -BEAM		SECTION_BEAM(resultant)
4 -SPRING		SECTION_BEAM(discrete)
6 –SOL_ORTH		SECTION_SOLID
8 –SPR_GENE		SECTION_BEAM(discrete)
9 –SH_ORTH		SECTION_SHELL
10-SH_COMP		SECTION_SHELL
11-SH_SANDW		SECTION_SHELL
12-SPR_PUL		SECTION_BEAM(discrete)
13-SPR_BEAM		SECTION_BEAM(discrete)
14-SOLID		SECTION_SOLID
32-SPR_PRE		SECTION_DISCRETE
35-STITCH		SECTION_DISCRETE
36-PREDIT		SECTION_BEAM(resultant)

CONVERT RADIOSS's SPRING PROPERTY(TYPE 4,8,12,13) INTO DYNA's

MATERIAL 196.

IF H=0, N1=0: LINEAR ELASTIC SPRING
K(DYNA) = K(RADIOSS)
D(DYNA) = C(RADIOSS)
IF H=0, N1>0: NONLINEAR ELASTIC SPRING
K(DYNA) = A(RADIOSS) (IF A(RADIOSS)=0.0, THEN
K(DYNA)=1.0)
C2(DYNA) = B(RADIOSS)/A(RADIOSS)
FLCID=GLCID = N1(RADIOSS)
HLCID = N2(RADIOSS)
DLE(DYNA) = D(RADIOSS)
IF H=1, N1>0:
K(DYNA) = K(RADIOSS)
A(RADIOSS) = 1.0 IS ASSUMED
C2(DYNA) = B(RADIOSS)
FLCID=GLCID = N1(RADIOSS)
HLCID = N2(RADIOSS)
DLE(DYNA) = D(RADIOSS)

CONVERT RADIOSS SPRING INTO DYNA BEAM.
IN ALL CASES, IF LENGTH OF SPRING IS FINITE, SET “SCOOR” OF
*SECTION_BEAM TO 2. IF LENGTH IS VERY SMALL(<=0.000001) OR ZERO,
SET “SCOOR” TO 0. IF RADIOSS DOES NOT HAVE INER VALUE, VPG SET
It AS 1.0.

8.3 MATERIAL PROPERTY:

RADIOSS(V4.4)	===>	DYNA(V970)
0 -VOID		9 -MAT_NULL
1 –ELAST		1 -MAT_ELASTIC
2 -PLAS_JOHNS or PLAS_ZERIL		98 -MAT_SIMPLIFIED_JOHNSON_COOK
3 –HYDPLA		10 -MAT_ELASTIC_PLASTIC_HYDRO
4 -HYD_JCOOK		15 -MAT_JOHNSON_COOK
6 -HYD_VISC		9 -MAT_NULL
10-LAW10		63 -MAT_CRUSHABLE_FOAM
14-COMPSO		2 -MAT_ORTHOTROPIC_ELASTIC
15-CHANG		2 -MAT_ORTHOTROPIC_ELASTIC
19-FABRI		130-MAT_SPECIAL_ORTHOTROPIC
21-DPRAG		57 -MAT_LOW_DENSITY_FOAM

NOTE:

There is no direct conversion of RADIOSS’s material type 21 to
DYNA’s *MAT_LOW_DENSITY_FOAM.
VPG uses the following example as DYNA’s material type 57 for seat foam.

```
$ UNITS:   kg, mm, ms, GPa
*MAT_LOW_DENSITY_FOAM
$   seat foam
571258.8200E-08      0.0045        2222          0.4          0.0          0.0          .05
0.0          0.0          0.0          0.0          0.0          0.0
*DEFINE_CURVE
2222              0          1.0          0.01          0.0          0.0              0
0.0                          0.0
0.0625                       0.119800001
0.122500002                  0.383390009
0.177499995                  0.377400011
0.234999999                  0.413399994
0.280699998                  0.444999993
0.344999999                  0.466100007
0.397000015                  0.497999996
0.444900006                  0.53670001
0.49000001                   0.561600029
0.528999984                  0.601000011
0.564999998                  0.653999984
0.597000003                  0.721000016
0.622500002                  0.818000019
0.644900024                  0.897000015
```

0.660000026	0.949999988
0.666999996	0.989000022
0.670000017	0.99000001
0.671999991	1.0
0.699000001	1.20000005
0.75	1.51999998
0.800000012	2.19000006
0.808000028	2.30299997

22-DAMA	81 -MAT_PLASTICITY_WITH_DAMAGE
23-LAW23	81 -MAT_PLASTICITY_WITH_DAMAGE
24-CONC	16 -MAT_PSEUDO_TENSOR
25-COMPSH	2 -MAT_ORTHOTROPIC_ELASTIC
27-PLAS_BRIT	81 -MAT_PLASTICITY_WITH_DAMAGE
28-HONEYCOMB	26 -MAT_HONEYCOMB

NOTE:

VPG set DYNA's value PR=0.0, SIGY=1.0, VF=0.2, MU=0.2, and ELFORM=1 in relative *SECTION_SOLID card. VPG create a *HOURGLASS card with IHQ=4, and put it into relative *PART's HGID field.
In RADIOSS, if the relative element property ID is 0 or the element property type is not TYPE 6, VPG will set the material directions 1,2,3 align with the global axes X,Y,Z, AOPT=2, (A1,A2,A3)=(1,0,0), (D1,D2,D3)=(0,1,0). Otherwise VPG will set AOPT=0, and map the RADIOSS 1,2,3 axes to the LS-DYNA b,c,a axes. The RADIOSS yield functions for the honeycomb material which have mu as the abscissa values must be converted into LS-DYNA curves with mu/(1+mu) as the abscissa values. In the above discussion, it is assumed that the angle PHI in the RADIOSS orthotropic solid property set data is set to zero. If PHI is nonzero, further thought must be given to the conversion.

32-HILL	3 -MAT_PLASTIC_KINEMATIC
33-FOAM_PLAS	53 -MAT_CLOSED_CELL_FOAM
34-BOLTZMAN	61 -MAT_KELVIN-MAXWELL_VISCOELASTIC
35-FOAM_VISC	76 -MAT_GENERAL_VISCOELASTIC
36-PLAS_TAB	24 -MAT_PIECEWISE_LINEAR_PLASTICITY
38-VISC_TAB	76 -MAT_GENERAL_VISCOELASTIC
40-KELVINMAX	76 -MAT_GENERAL_VISCOELASTIC
42-OGDEN	77 -MAT_OGDEN_RUBBER
43-HILL_TAB	24 -MAT_PIECEWISE_LINEAR_PLASTICITY
44-COWPER	98 -MAT_SIMPLIFIED_JOHNSON_COOK
48-ZHAO	98 -MAT_SIMPLIFIED_JOHNSON_COOK
49–STEINB	10 -MAT_ELASTIC_PLASTIC_HYDRO
50-LAW50	26 -MAT_HONEYCOMB
52-GURSON	98 -MAT_SIMPLIFIED_JOHNSON_COOK
53-LAW53	26 -MAT_HONEYCOMB
54–LAW54	1 -MAT_ELASTIC

8.4 BOUNDARY AND LOAD CONDITION:

RADIOSS (V4.4) ===>	DYNA (V970)
BCS	BOUNDARY_SPC_SET
CLOAD	LOAD_NODE_SET
GRAVITY	LOAD_BODY
INITIAL VELOCITY	INITIAL_VELOCITY
IMPOSED DISPLACEMENT	BOUNDARY_PRESCRIBED_MOTION_SET
IMPOSED VELOCITY	BOUNDARY_PRESCRIBED_MOTION_SET
IMPOSED ACCELERATION	BOUNDARY_PRESCRIBED_MOTION_SET
MPC	CONSTRAINED_LINEAR_LOCAL
PLOAD	LOAD_SEGMENT_SET

8.5 CONTACT AND RIGIDWALL:

Delete RIGIDWALLs which do not have nodes.

RADIOSS(V4.4)	===>	DYNA(V970)
TYPE2 -TIED		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE3 -SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE5 -SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE6 -SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE7 -SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE8 -SLIDE		CONTACT_DRAWBEAD
TYPE10-TIED/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE11-SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE14-SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
TYPE15-SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
LAGMUL TYPE2 -TIED		CONTACT_AUTOMATIC_SINGLE_SURFACE
LAGMUL TYPE7 -SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
LAGMUL TYPE16-SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE
LAGMUL TYPE17-SLIDE/VOID		CONTACT_AUTOMATIC_SINGLE_SURFACE

NOTE:

Or create a new *CONTACT_AUTOMATIC_SINGLE_SURFACE to replace all RADIOSS's contact.
Set all *CONTACT_AUTOMATIC_SINGLE_SURFACE's SOFT=1, IGAP=2, IGNORE=1.
If RADIOSS data does not have FS, VPG will set it as 0.2. Moreover,
set FD as 0.2.

RIGIDWALL RIGIDWALL_GEOMETRIC

8.6. AIRBAG(MONITORED VOLUMES):

MONITORED VOLUMES AIRBAG_SIMPLE_AIRBAG_MODEL
If this MONITORED VOLUMES is a tire model, we suggest use
AIRBAG_SIMPLE_PRESSURE_VOLUME instead of AIRBAG_SIMPLE_AIRBAG_MODEL.

REPLACE THE ORIGINAL MATERIAL WITH *MAT_FABRIC.

8.7 SEATBELT:

CONVERT RADIOSS'S MATERIAL TYPE 1 WHICH HAS VERY SMALL YOUNG'S MODULUS INTO DYNA'S *MAT_FABRIC.

8.8 OTHERS:

Delete all other cards that can't be converted to DYNA.

RADIOSS (V4.4)	===>	DYNA (V970)
FUNCTION		DEFINE_CURVE
FRAME		DEFINE_COORDINATE

NOTE:

If you don't have RADIOSS ENGINE file, VPG will set up DYNA *CONTROL
cards for you as follows.

```
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*CONTROL_HOURGLASS
$         ihq            qh
              4           .1
*CONTROL_ACCURACY
$         osu           inn      pidosu
              0             2           0
*CONTROL_BULK_VISCOSITY
$          q2            q1         type
       1.500     0.06000             1
*CONTROL_CONTACT
$     slsfac      rwpnal      islchk      shlthk      penopt      thkchg       orien      enmass
    0.10000       1.000           2           0           1           1           2           0
$     usrstr      usrfrc      nsbcs      interm       xpene       ssthk        ecdt     tiedprj
             0           0          10           0       4.000           0           0           0
```

```
$       sfric        dfric         edc          vfc           th        th_sf      pen_sf
          0.0          0.0         0.0          0.0          0.0          0.0         0.0
$     ignore       frceng
             1            0
*CONTROL_ENERGY
$        hgen         rwen       slnten        rylen
             2            2            2            2
*CONTROL_OUTPUT
$       npopt       neecho       nrefup       iaccop        opifs      ipnint      ikedit      iflush
             0            0            0            0          0.0           0         200        5000
$       iprtf
             0
*CONTROL_SHELL
$     wrpang       itrist         irnxx       istupd       theory          bwc        miter         proj
     20.000            1           -1            0            2            2            1            0
$    rotascl       intgrd       lamsht
          0.0            0            0
*CONTROL_TERMINATION
$     endtim       endcyc        dtmin       endeng       endmas
      0.000            0          0.0          0.0          0.0
*CONTROL_TIMESTEP
$     dtinit       tssfac         isdo       tslimt         dt2ms         lctm        erode        ms1st
          0.0      0.90000            0          0.0 -1.100E-3            0            0            0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

I. If you don't have RADIOSS ENGINE file, VPG will set up DYNA *DATABASE cards for you as following.

```
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*DATABASE_ABSTAT
$            dt
         0.5
*DATABASE_GLSTAT
$            dt
         0.5
*DATABASE_MATSUM
$            dt
        1.000
*DATABASE_NODOUT
$            dt
         0.5
*DATABASE_RCFORC
$            dt
         0.5
*DATABASE_RWFORC
$            dt
         0.5
*DATABASE_SECFORC
$            dt
         0.5
*DATABASE_ELOUT
$            dt
         0.5
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*DATABASE_BINARY_D3PLOT
$    dt/cycl         lcdt         beam         npltc
       5.000            0            0            0
*DATABASE_BINARY_D3DUMP
$    dt/cycl
     100000.
*DATABASE_BINARY_RUNRSF
$    dt/cycl
      1000.
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
```

9. CONVERTING FROM DYNA TO RADIOSS

9.1 ELEMENT:

Delete PLOTEL, *CONSTRAINED_INTERPOLATION element.

DYNA (V970)	===>	RADIOSS (V4.4)
BEAM		BEAM
DISCRETE		SPRING
Convert DYNA's VID into RADIOSS's third node.		
INERTIA		MASS
MASS		MASS
SHELL		SHELL
SOLID		SOLID
T-SHELL		SOLID
CONSTRAINED_NODAL_RIGID_BODY		RBODY
CONSTRAINED_NODE_SET		RBODY
CONSTRAINED_SPOTWELD		RIVET
JOINT		JOINT
SEATBELT		BEAM

9.2 ELEMENT PROPERTY:

Delete SPH, POINT_SOURCE, POINT_SOURCE_MIXTURE section.
Convert DYNA's spring stiffness into RADIOSS's element property.

DYNA (V970)	===>	RADIOSS (V4.4)
SECTION_BEAM (integrated)		3 -BEAM
SECTION_BEAM (resultant)		3 -BEAM
SECTION_BEAM (discrete)		3 -BEAM
SECTION_BEAM (truss)		3 -BEAM
SECTION_BEAM (spotweld)		3 –BEAM
SECTION_BEAM (2D shells)		3 -BEAM
SECTION_DISCRETE		4 -SPRING
SECTION_SHELL		1 -SHELL
SECTION_SHELL_ALE		1 -SHELL
SECTION_SOLID		14-SOLID
SECTION_SOLID_ALE		14-SOLID
SECTION_TSHELL		14-SOLID
SECTION_SEATBELT		3 -BEAM

9.3 MATERIAL PROPERTY:

DYNA (V970)	===>	RADIOSS (V4.4)
1 -MAT_ELASTIC		1 -ELAST
1 -MAT_ELASTIC_FLUID		1 -ELAST
2 -MAT_ORTHOTROPIC_ELASTIC		14-COMPSO
2 -MAT_ANISOTROPIC_ELASTIC		1 -ELAST
3 -MAT_PLASTIC_KINEMATIC		2 -PLAS_JOHNS
4 -MAT_ELASTIC_PLASTIC_THERMAL		2 -PLAS_JOHNS
5 -MAT_SOIL_AND_FOAM		33-FOAM_PLAS
6 -MAT_VISCOELASTIC		34-BOLTZMAN
7 -MAT_BLATZ-KO_RUBBER		1 -ELAST
8 -MAT_HIGH_EXPLOSIVE_BURN		1 -ELAST
9 -MAT_NULL		6 -HYD_VISC
10 -MAT_ELASTIC_PLASTIC_HYDRO		3 -HYDPLA
10 -MAT_ELASTIC_PLASTIC_HYDRO_SPALL		3 -HYDPLA
11 -MAT_STEINBERG		1 -ELAST
11 -MAT_STEINBERG_LUND		1 -ELAST
12 -MAT_ISOTROPIC_ELASTIC_PLASTIC		2 -PLAS_JOHNS
13 -MAT_ISOTROPIC_ELASTIC_FAILURE		2 -PLAS_JOHNS
14 -MAT_SOIL_AND_FOAM_FAILURE		33-FOAM_PLAS

15 -MAT_JOHNSON_COOK 4 -HYD_JCOOK
16 -MAT_PSEUDO_TENSOR 24-CONC
17 -MAT_ORIENTED_CRACK 2 -PLAS_JOHNS
18 -MAT_POWER_LAW_PLASTICITY 2 -PLAS_JOHNS
19 -MAT_STRAIN_RATE_DEPENDENT_PLASTICITY 2 -PLAS_JOHNS
20 -MAT_RIGID 1 -ELAST
21 -MAT_ORTHOTROPIC_THERMAL 14-COMPSO
22 -MAT_COMPOSITE_DAMAGE 14-COMPSO
23 -MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC
14-COMPSO
24 -MAT_PIECEWISE_LINEAR_PLASTICITY 36-PLAS_TAB
25 -MAT_GEOLOGICAL_CAP_MODEL 24-CONC
26 -MAT_HONEYCOMB 2 -PLAS_JOHNS
27 -MAT_MOONEY-RIVLIN_RUBBER 1 -ELAST
28 -MAT_RESULTANT_PLASTICITY 2 -PLAS_JOHNS
29 -MAT_FORCE_LIMITED 2 -PLAS_JOHNS
30 -MAT_CLOSED_FORM_SHELL_PLASTICITY 2 -PLAS_JOHNS
31 -MAT_FRAZER_NASH_RUBBER_MODEL 1 -ELAST
32 -MAT_LAMINATED_GLASS 2 -PLAS_JOHNS
33 -MAT_BARLAT_ANISOTROPIC_PLASTICITY 2 -PLAS_JOHNS
33 -MAT_BARLAT_YLD96 2 -PLAS_JOHNS
34 -MAT_FABRIC 14-COMPSO
35 -MAT_PLASTIC_GREEN-NAGHDI_RATE 2 -PLAS_JOHNS
36 -MAT_3-PARAMETER_BARLAT 2 -PLAS_JOHNS
37 -MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC
2 -PLAS_JOHNS
38 -MAT_BLATZ-KO_FOAM 33-FOAM_PLAS
39 -MAT_FLD_TRANSVERSELY_ANISOTROPIC 2 -PLAS_JOHNS
40 -MAT_NONLINEAR_ORTHOTROPIC 14-COMPSO
41 -MAT_USER_DEFINED_MATERIAL_MODELS 1 -ELAST
51 -MAT_BAMMAN 2 -PLAS_JOHNS
52 -MAT_BAMMAN_DAMAGE 2 -PLAS_JOHNS
53 -MAT_CLOSED_CELL_FOAM 33-FOAM_PLAS
54 -MAT_ENHANCED_COMPOSITE_DAMAGE 14-COMPSO
57 -MAT_LOW_DENSITY_FOAM 33-FOAM_PLAS
58 -MAT_LAMINATED_COMPOSITE_FABRIC 14-COMPSO
59 -MAT_COMPOSITE_FAILURE_SHELL_MODEL 14-COMPSO
59 -MAT_COMPOSITE_FAILURE_SOLID_MODEL 14-COMPSO
60 -MAT_ELASTIC_WITH_VISCOSITY 1 -ELAST
61 -MAT_KELVIN-MAXWELL_VISCOELASTIC 34-BOLTZMAN
62 -MAT_VISCOUS_FOAM 33-FOAM_PLAS
63 -MAT_CRUSHABLE_FOAM 10-LAW10
64 -MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY
2 -PLAS_JOHNS
65 -MAT_MODIFIED_ZERILLI_ARMSTRONG 2 -PLAS_JOHNS
66 -MAT_LINEAR_ELASTIC_DISCRETE_BEAM 1 –ELAST
67 -MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM 1 –ELAST
68 -MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM 2 -PLAS_JOHNS
69 -MAT_SID_DAMPER_DISCRETE_BEAM 1 –ELAST
70 -MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM
1 –ELAST
71 -MAT_CABLE_DISCRETE_BEAM 1 -ELAST
72 -MAT_CONCRETE_DAMAGE 24-CONC
73 -MAT_LOW_DENSITY_VISCOUS_FOAM 33-FOAM_PLAS
74 -MAT_ELASTIC_SPRING_DISCRETE_BEAM 1 -ELAST
75 -MAT_BILKHU/DUBOIS_FOAM 33-FOAM_PLAS
76 -MAT_GENERAL_VISCOELASTIC 35-FOAM_VISC
77 -MAT_HYPERELASTIC_RUBBER 42-OGDEN
77 -MAT_OGDEN_RUBBER 42-OGDEN
78 -MAT_SOIL_CONCRETE 10-LAW10
79 -MAT_HYSTERETIC_SOIL 24-CONC
80 -MAT_RAMBERG_OSGOOD 1 -ELAST
81 -MAT_PLASTICITY_WITH_DAMAGE 22-DAMA
83 -MAT_FU_CHANG_FOAM 33-FOAM_PLAS
86 -MAT_ORTHOTROPIC_VISCOELASTIC 14-COMPSO
87 -MAT_CELLULAR_RUBBER 1 -ELAST
88 -MAT_MTS 1 -ELAST
89 -MAT_PLASTICITY_POLYMER 2 -PLAS_JOHNS
90 -MAT_ACOUSTIC 1 -ELAST
94 -MAT_INELASTIC_SPRING_DISCRETE_BEAM 1 -ELAST

96 -MAT_BRITTLE_DAMAGE	24-CONC
98 -MAT_SIMPLIFIED_JOHNSON_COOK	2 -PLAS_JOHNS
100-MAT_SPOTWELD	1 -ELAST
103-MAT_ANISOTROPIC_VISCOPLASTIC	33-FOAM_PLAS
104-MAT_DAMAGE_1	22-DAMA
105-MAT_DAMAGE_2	22-DAMA
116-MAT_COMPOSITE_LAYUP	14-COMPSO
117-MAT_COMPOSITE_MATRIX	1 -ELAST
118-MAT_COMPOSITE_DIRECT	1 –ELAST
123-MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY	36-PLAS_TAB
124-MAT_PLASTICITY_COMPRESSION_TENSION	2 -PLAS_JOHNS
126-MAT_MODIFIED_HONEYCOMB	2 -PLAS_JOHNS
127-MAT_ARRUDA_BOYCE_RUBBER	42-OGDEN
128-MAT_HEART_TISSUE	1 -ELAST
129-MAT_LUNG_TISSUE	1 -ELAST
130-MAT_SPECIAL_ORTHOTROPIC	19-FABRI
OTHER MATERIAL TYPE	1 -ELAST

9.4 BOUNDARY AND LOAD CONDITION:

DYNA (V970)	===>	RADIOSS (V4.4)
BOUNDARY_PRESCRIBED_MOTION(displacement)		IMPOSED DISPLACEMENT
BOUNDARY_PRESCRIBED_MOTION(velocity)		IMPOSED VELOCITY
BOUNDARY_PRESCRIBED_MOTION(acceleration)		IMPOSED ACCELERATION
BOUNDARY_SPC		BCS
CONSTRAINED_LINEAR		MPC
INITIAL_VELOCITY		INITIAL VELOCITY
LOAD_NODE (Deleted when DOF is 4 or 8.)		CLOAD
LOAD_SEGMENT		PLOAD
LOAD_SHELL		PLOAD

9.5 CONTACT AND RIGIDWALL:

VPG deletes CONTACT_ENTITY, CONTACT_INTERIOR, CONTACT_1D, CONTACT_2D_OPTIONS, CONTACT_GEBOD_OPTION, CONTACT_RIGID_SURFACE, CONTACT_AUTO_MOVE, CONTACT_COUPLING.

DYNA (V970)	===>	RADIOSS (V4.4)
CONTACT_AIRBAG_SINGLE_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_GENERAL		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_GENERAL_INTERIOR		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_NODES_TO_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_SINGLE_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_SURFACE_TO_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK		TYPE7 -SLIDE/VOID
CONTACT_CONSTRAINT_NODES_TO_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_CONSTRAINT_SURFACE_TO_SURFACE		TYPE7 -SLIDE/VOID
CONTACT_DRAWBEAD		TYPE8 –SLIDE

CONTACT_DRAWBEAD_INITIALIZE	TYPE8 –SLIDE
CONTACT_ERODING_NODES_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_ERODING_SINGLE_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_ERODING_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_FORCE_TRANSDUCER_PENALTY	TYPE7 -SLIDE/VOID
CONTACT_FORCE_TRANSDUCER_CONSTRAINT	TYPE7 -SLIDE/VOID
CONTACT_FORMING_NODES_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_FORMING_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_NODES_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_NODES_TO_SURFACE_INTERFERENCE	TYPE7 -SLIDE/VOID
CONTACT_ONE_WAY_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE	TYPE7 -SLIDE/VOID
CONTACT_RIGID_NODES_TO_RIGID_BODY	TYPE7 -SLIDE/VOID
CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY	TYPE7 -SLIDE/VOID
CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY	TYPE7 -SLIDE/VOID
CONTACT_SINGLE_EDGE	TYPE7 -SLIDE/VOID
CONTACT_SINGLE_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_SLIDING_ONLY	TYPE7 -SLIDE/VOID
CONTACT_SLIDING_ONLY_PENALTY	TYPE7 -SLIDE/VOID
CONTACT_SPOTWELD	TYPE7 -SLIDE/VOID
CONTACT_SPOTWELD_WITH_TORSION	TYPE7 -SLIDE/VOID
CONTACT_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_SURFACE_TO_SURFACE_THERMAL	TYPE7 -SLIDE/VOID
CONTACT_SURFACE_TO_SURFACE_INTERFERENCE	TYPE7 -SLIDE/VOID
CONTACT_TIEBREAK_NODES_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_TIEBREAK_NODES_ONLY	TYPE7 -SLIDE/VOID
CONTACT_TIEBREAK_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_TIED_NODES_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_TIED_NODES_TO_SURFACE_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SURFACE_TO_SURFACE	TYPE7 -SLIDE/VOID
CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET	TYPE7 -SLIDE/VOID
CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE	TYPE7 -SLIDE/VOID
RIGIDWALL	RWALL

9.6 OTHERS:

Delete all other cards that can't be converted to RADIOSS.

DYNA (V970)	===>	RADIOSS (V4.4)
DEFINE_COORDINATE		FRAME
DEFINE_CURVE		FUNCTION

10. CONVERTING FROM NASTRAN TO DYNA

10.1 ELEMENT:

When some RBE2 share the common node, merge them into one RBE2.
Delete CGAP elements.

NASTRAN	===>	DYNA (V970)
CBAR		BEAM
RBE2		If NASTRAN's DOF is 123456, convert it into DYNA's *CONSTRAINED_NODAL_RIGID_BODY. Otherwise, convert it into DYNA's *CONSTRAINED_NODE_SET.
SHELL		SHELL
SOLID		SOLID
CELAS2		DISCRETE Create a *DEFINE_SD_ORIENTATION for spring direction.
CELAS1		DISCRETE Create a *DEFINE_SD_ORIENTATION for spring direction.
RBAR		CONSTRAINED_SPOTWELD
CONM2		MASS & INERTIA
CBEAM		BEAM
CROD		BEAM
CTUBE		BEAM
ISOPARAMETRIC SHELL		SHELL
RBE3		CONST. INTERPOLATION

10.2 ELEMENT PROPERTY:

VPG deletes PGAP property. VPG converts spring stiffness into DYNA material.

NASTRAN	===>	DYNA (V970)
PSHELL		SECTION_SHELL
PSOLID		SECTION_SOLID
PBAR		SECTION_BEAM (resultant)
PBEAM		SECTION_BEAM (resultant)
PELAS		SECTION_DISCRETE
PROD		SECTION_BEAM (truss)
PTUBE		SECTION_BEAM (integrated)

10.3 MATERIAL PROPERTY:

NASTRAN	===>	DYNA (V970)
MAT1		1 -MAT_ELASTIC

10.4 BOUNDARY AND LOAD CONDITION:

NASTRAN	===>	DYNA(V970)
SPC		BOUNDARY_SPC_SET
NODE FORCE		LOAD_NODE_SET
SHELL PRESSURE		LOAD_SHELL_SET
INITIAL VELOCITY		INITIAL_VELOCITY

10.5 OTHERS:

VPG converts NODE SET, ELEMENT SET.

NASTRAN	===>	DYNA (V970)
LOAD CURVE		DEFINE_CURVE
LOCAL COORDINATE		DEFINE_COORDINATE_SYSTEM

11. CONVERTING FROM DYNA TO NASTRAN

11.1 ELEMENT:

Delete SEATBELT & JOINT elements.

DYNA (V970)	===>	NASTRAN
BEAM		CBAR, CBEAM, CROD According to it's element property.
CONSTRAINED_NODAL_RIGID_BODY		RBE2
T-SHELL		SOLID
SHELL		SHELL
SOLID		SOLID
DISCRETE		CELAS1 Create a LCS for spring direction.
MASS		CONM2
CONSTRAINED_SPOTWELD		RBAR
INERTIA		CONM2
CONSTRAINED_NODE_SET		RBE2

11.2 ELEMENT PROPERTY:

VPG seletes SEATBELT section and converts spring stiffness into element property.

DYNA (V970)	===>	NASTRAN
SECTION_BEAM(integrated)		PBAR According to it's ELFORM and CST.
SECTION_BEAM (resultant)		PBAR
SECTION_BEAM (discrete)		PBAR
SECTION_BEAM (truss)		PROD
SECTION_BEAM (spotweld)		PBAR
SECTION_DISCRETE		PELAS
SECTION_SHELL		PSHELL
SECTION_SOLID		PSOLID
SECTION_SOLID_ALE		PSOLID
SECTION_TSHELL		PSOLID

11.3 MATERIAL PROPERTY:

All material MAT1.

11.4 BOUNDARY AND LOAD CONDITION:

DYNA (V970)	===>	NASTRAN

BOUNDARY_SPC_NODE	SPC
BOUNDARY_SPC_SET	SPC
INITIAL_VELOCITY	INITIAL V ELOCITY
INITIAL_VELOCITY_NODE	INITIAL VELOCITY
INITIAL_VELOCITY_GENERATION	INITIAL VELOCITY
LOAD_NODE_POINT	NODAL FORCE
LOAD_NODE_SET	NODAL FORCE
LOAD_SHELL_ELEMENT	SHELL PRESSURE
LOAD_SHELL_SET	SHELL PRESSURE

11.5 OTHERS:

Convert NODE SET, ELEMENT SET. Delete all other cards that can't be converted.

DYNA (V970)	===>	NASTRAN
DEFINE_COORDINATE_NODES		LOCAL COORDINATE
DEFINE_COORDINATE_SYSTEM		LOCAL COORDINATE
DEFINE_COORDINATE_VECTOR		LOCAL COORDINATE
DEFINE_CURVE		LOAD CURVE

12. CONVERTING FROM RADIOSS TO NASTRAN

12.1 ELEMENT:

RADIOSS (V4.4)	===>	NASTRAN
BEAM		CBAR
JOINT		RBE2
MASS		CONM2
QUAD		SHELL
RBODY		RBE2
RIVET		RBAR
RLINK		RBE2
SHELL		SHELL
SHEL16		SOLID
SOLID		SOLID
SPRING		CELAS1
	(The third node is converted into an LCS.)	
TRUSS		CROD
XELEM		RBE2

12.2 ELEMENT PROPERTY:

Delete TYPE0(VOID), TYPE5(RIVET), TYPE28(NSTRAND) property.

RADIOSS (V4.4)	===>	NASTRAN
1 -SHELL		PSHELL
2 -TRUSS		PROD
3 -BEAM		PBAR
4 -SPRING		PELAS
6 -ORTHOTROPIC SOLID		PSOLID
8 -GENERAL SPRING		PELAS
9 -ORTHOTROPIC SHELL		PSHELL
10-COMP. SHELL		PSHELL
11-COMP. SHELL		PSHELL
12-3-NODES SPRING		PELAS
13-BEAM TYPE SPRING		PELAS

14-GENERAL SOLID	PSOLID
32-PRETENSIONNER SPRING	PELAS
33-JOINT TYPE SPRING	PELAS
35-AIRBAG STITCH SPRING	PELAS
36-PREDIT SPRING	PELAS

12.3 MATERIAL PROPERTY:

All materials	MAT1

12.4 BOUNDARY AND LOAD CONDITION:

VPG deletes PRESSURE LOAD.

RADIOSS (V4.4)	===>	NASTRAN
BCS		SPC
CLOAD		NODE FORCE
INITIAL VELOCITY		INITIAL VELOCITY

12.5 OTHERS:

VPG deletes all other cards that can't be converted.

RADIOSS (V4.4)	===>	NASTRAN
FUNCTION		LOAD CURVE
FRAME		LOCAL COORDINATE

13. CONVERTING FROM NASTRAN TO RADIOSS

13.1 ELEMENT:

Delete RBE3, CGAP element.

NASTRAN	===>	RADIOSS (V4.4)
CBAR		BEAM
RBE2		RBODY
SHELL		SHELL
SOLID		SOLID
CELAS2		SPRING
	(Create a node for spring direction.)	
CELAS1		SPRING
	(Create a node for spring direction.)	
RBAR		RIVET
CONM2		MASS
CBEAM		BEAM
CROD		TRUSS
CTUBE		BEAM
ISOPARAMETRIC SHELL		SHELL

13.2 ELEMENT PROPERTY:

VPG deletes PGAP properties.

NASTRAN	===>	RADIOSS (V4.4)

PSHELL	1 -SHELL
PSOLID	14-GENERAL_SOLID
PBAR	3 -BEAM
PBEAM	3 -BEAM
PELAS	4 -SPRING
PROD	2 -TRUSS
PTUBE	3 -BEAM

13.3 MATERIAL PROPERTY:

NASTRAN	===>	RADIOSS (V4.4)
MAT1		1 -ELASTIC

13.4 BOUNDARY AND LOAD CONDITION:

NASTRAN	===>	RADIOSS (V4.4)
SPC		BCS
NODE FORCE		CLOAD
SHELL PRESSURE		PLOAD
INITIAL VELOCITY		INITIAL VELOCITY

13.5 OTHERS:

VPG deletes all other cards that can't be converted.

NASTRAN	===>	RADIOSS (V4.4)
LOAD CURVE		FUNCTION
LOCAL COORDINATE SYSTEM		FRAME